MODELEUR
User's Guide

MODELEUR version 1.0a07
June 2000

Copyright © 1999-2000 INRS

Table of contents

Chapter 1 General description 1-1
Infrastructure 1-1
Hardware and software requirements 1-1
Installation 1-2
Physical installation on disk 1-2
Installation in the programming environment 1-3
Launching 1-4
Registration procedure 1-4
Description of the user interface 1-5
Main window 1-5
Project Menu 1-6
Modeler Menu 1-7
Simulation Menu 1-8
Display Menu 1-8
Tools Menu 1-8
Window Menu 1-8
Help Menu 1-9
Display windows 1-9
Create a window 1-9
Rearrange the windows 1-10
Close all the windows 1-10
Dialog boxes 1-10

Chapter 2 Work procedure 2-1
Import terrain data 2-1
Construct the numerical terrain model 2-1
Conduct a simulation 2-3

Chapter 3 Project management 3-1
Create a new project 3-1
Identify a new project 3-4
Specify the directory for the project 3-4
Base a project on existing data 3-4
Georeference and positioning 3-5
Open an existing project 3-6
Save the current project 3-7
Save As...to save a project under a different name 3-8
Close the project 3-9
Modify the parameters of the current project 3-10
Import an entity 3-11
Identify an entity to be imported 3-14
Select the files to be imported 3-14
Select a filter 3-14
Select the links for an entity to be imported 3-15
Export an entity 3-16
Identify an entity to be exported 3-17
Specify the file(s) to be exported 3-17
Select a filter 3-18
Handling the project's entities 3-18
Display a sorted list of the entities 3-20
Display a list of the entities on selected criteria 3-20
View the links between entities 3-21
Modify the label and/or the description of an entity 3-22

Delete an entity from the project 3-22
Display information on the characteristics of the entity 3-24

Chapter 4 Controlling displays 4-1
Managing the graphic elements 4-1
Opening a window 4-3
Closing a window 4-3
Activating a window 4-3
Renaming a window 4-4
Changing the cursor for a window 4-5
Displaying a graphic element [screen] 4-6
Displaying a graphic element [printer] 4-6
Deleting a graphic element 4-7
Changing the order of display of the elements 4-8
Moving an element to another window 4-8
Modifying the display parameters of an element 4-9
Saving a graphic element 4-10
Loading a graphic element previously saved 4-11
Displaying the legend associated to a graphic element 4-12
Creating a group 4-13
Closing a group 4-14
Renaming a group 4-14
Adding a window to a group 4-15
Extracting a window from a group 4-15
Controlling the view 4-16
Zooming in 4-17
Zooming out 4-18
Re-centering the view 4-19
Making a rotation 4-20
Returning to the preceding view 4-21
Returning to the initial view 4-21
Printing the content of a window 4-21
Specify printing destination 4-22
Specify image printing size 4-23
Specify printing setup 4-24

Chapter 5 Data display 5-1
Data display 5-1
Colour parameters setup 5-2
Display a mesh 5-3
Display element numbers 5-4
Display reduced elements 5-5
Display isolines 5-5
Specify the number of isolines 5-7
Display isosurfaces 5-8
Specify the number of isosurfaces 5-9
Display vectors 5-11
Specify vector body parameters 5-13
Specify the vector point characteristics 5-13
Display the mesh nodes 5-14
Display node numbers 5-15
Display profiles 5-15
Specify colours for profile elements 5-17
Specify profile title and axes names 5-17
Specify types of lines and points for the profile 5-18
Specify the X axis parameters for the profile 5-19

Specify the Y axis parameters for the profile 5-19
Specify the calculation parameters for the profile 5-20
Cartographic dressing 5-21
Cartographic dressing management 5-21
Create a cartographic dressing 5-22
Load a cartographic dressing 5-22
Save a cartographic dressing 5-23
Save As...a cartographic dressing 5-24
Close a cartographic dressing 5-24
Element management 5-25
Create an element 5-26
Modify an element 5-26
Delete an element 5-33
Copy an element 5-33
Cut an element 5-33
Paste an element 5-34

Chapter 6 Data preparation 6-1
Specification tool : the partition 6-2
Create a new partition 6-3
Load an existing partition 6-4
Save a partition 6-4
Save a partition under a different name (Save As) 6-5
Close a partition 6-6
Create a sub-domain 6-7
Move an element of a partition 6-7
Erase a sub-domain from a partition 6-8
Merge two elements of a partition 6-8
Split an element of a partition 6-9
Delete a vertex 6-9
Generate a mesh 6-10
Specify the mesh parameters 6-11
Global parameters 6-11
Default parameters 6-13
Parameters specific to a vertex 6-14
Generate a mesh on a surface 6-16
Assemble the global mesh 6-17
Projection of the topography 6-19
Import the sources of topography data 6-20
Build the topography partition 6-21
Specify the topography parameters 6-22
Default parameters 6-22
Surface parameters 6-23
Assemble the global topography 6-25
Specify the boundary conditions 6-27
Specify the boundary conditions parameters 6-28
Generate the boundary conditions 6-30
Specify the wind characteristics 6-31
Specify the wind parameters 6-32
Default parameters 6-32
Surface parameters 6-33
Generate a vector field - wind 6-34
Specify the substrate characteristics 6-35
Determine the classification of the substrates 6-37
Import substrate terrains (optional) 6-37

Build a substrate partition 6-38
Specify the substrate parameters 6-39
Default parameters 6-39
Surface parameters 6-40
Assemble the substrate on a designated mesh 6-42
Specify the ice characteristics 6-44
Specify the ice parameters 6-44
Default parameters 6-45
Surface parameters 6-45
Assemble a field for ice 6-46
Specify the other friction coefficients 6-47
Specify the Manning friction parameters 6-48
Default parameters 6-48
Surface parameters 6-49
Assemble a scalar field - Manning friction 6-49

Chapter 7 Conducting a simulation 7-1
Specify the characteristics of a simulation 7-1
Zone of the control buttons 7-1
Create a new simulation 7-2
Load a simulation 7-3
Save the simulation 7-3
Save the simulation under a different name (Save As...) 7-4
Close the simulation 7-5
Zone of specifications display 7-5
Edit a command 7-6
Remove a command 7-6
Move a command 7-7
Simulation parameters zone 7-7
Print detail level 7-8
Formulation 7-10
Mesh 7-11
Global properties 7-12
Topography 7-14
Friction 7-16
Initial conditions 7-17
Boundary conditions 7-20
Wind 7-21
Resolution scheme 7-22
Results 7-22
Launch and monitor a simulation 7-23
Recover the results 7-25
Validate the results obtained 7-25
Content of a simulation directory 7-26

Chapter 8 Using the tools 8-1
Data probe 8-1
Select the type of coordinate system 8-2
On-click - continuous mode 8-3
Add a field to the data probe 8-3
Remove a field from the data probe 8-5
Calculator 8-6
Results zone 8-7
Command line 8-7
Numerical keyboard 8-7
List of variables 8-8

The functions of the calculator 8-9
Scientific mode 8-9
Statistical mode 8-9
Integral mode 8-10
User functions mode 8-10
Load a field 8-11
Save a field 8-12
Function editor 8-13
Create a new function 8-14
Load an existing function 8-14
Save a function 8-15
Save As... a function 8-16
Close a function 8-16
Delete a function 8-17
Compile a function 8-17
Cut selected code 8-18
Copy selected code 8-18
Paste code 8-19
Cancel an operation 8-19
Projection tool 8-19
Specify the source field 8-20
Specify a destination field 8-21
Profile mode 8-22
Profile management 8-22
Create a profile 8-23
Load a profile 8-23
Save a profile 8-23
Save As... a profile 8-24
Close a profile 8-25
Profile edition 8-25
Display the parameters window 8-26
Create the polyline of the profile 8-28
Move the elements of the profile 8-28
Erase the polyline of the profile 8-29
Split a segment in two 8-29
Eliminate a vertex from a polyline 8-29

Chapter 9 The data model 9-1

Field 9-3
Scalar field 9-3
Scalar field - topography 9-4
Scalar field - surface level 9-4
Scalar field - depth 9-5
Scalar field - velocity modulus 9-6
Scalar field - Froude number 9-6
Scalar field - shear stress velocity 9-6
Scalar field - turbulent viscosity 9-7
Scalar field - numerical viscosity 9-7
Scalar field - local mass balance 9-7
Scalar field - relative mass balance 9-8
Scalar field - residuals of specific discharge in X 9-8
Scalar field - residuals of specific discharge in Y 9-9
Scalar field - residuals of water level 9-9
Scalar field - error of water level 9-9
Scalar field - error of specific discharge in X 9-10

Scalar field - error of specific discharge in Y 9-10
Scalar field - Manning friction 9-10
Scalar field - substrate friction 9-11
Scalar field - topography terrain 9-11
Vector field 9-12
Vector field - velocity 9-12
Vector field - wind 9-13
Field - N values vector 9-13
Field - ice 9-14
Field - percentage of substrates 9-14
Substrate classes 9-15
Boundary conditions 9-16
Reference event 9-17
Function 9-17
System function 9-17
User function 9-17
Partition mesh 9-17
Mesh 9-17
Partition 9-18
Mesh partition 9-19
Topography partition 9-20
Boundary conditions partition 9-21
Substrates partition 9-22
Wind partition 9-22
Ice partition 9-23
Manning friction partition 9-24
Coordinate system 9-25
Data points 9-25
Hydrodynamic simulation 9-26
Topographic terrain 9-26
Substrates terrain 9-27

Chapter 10 Data formats 10-1
Raw topography data format 10-1
Data points format 10-2
Mesh format 10-2
Scalar field format 10-3
Vector field format 10-4
Coordinate system format 10-4
Cartographic coordinate systems 10-4
Geographic coordinate systems 10-6
Cartesian coordinate systems 10-6
Function format 10-7
Substrate classes format 10-7
Mesh partition format 10-8
Topography partition format 10-9
Wind partition format 10-10
Substrate partition format 10-11
Ice partition format 10-13
Friction partition format 10-14

Chapter 11 Programming language for the calculator 11-1
Syntax 11-1
Key words 11-2
Symbols 11-2
Basic types 11-3

Expression 11-3
Arithmetic expression 11-3
Logical expression 11-4
Operators 11-4
() 11-5
.x , .y 11-6
{} 11-6
Control structures 11-6
if - else if - else - end if 11-7
if - else 11-7
for - end for 11-8
while - end while 11-8
with - end with 11-9
Functions 11-9
Definition of a user function 11-9
Function request 11-10
Predefined functions 11-10
abs 11-11
fix 11-12
int 11-12
sqrt 11-12
min 11-13
max 11-13
sin 11-14
cos 11-15
tan 11-15
asin 11-16
acos 11-16
atan 11-16
log 11-17
ln 11-17
exp 11-18
ddx, ddy 11-18
intg 11-18
savefield 11-19

Chapter 1 General description

MODELEUR is a software combining the features of a Geographical Information System (GIS) applied to river hydraulics, with powerful capacities of pre and post processing of the various finite element components. MODELEUR is coupled with a Finite Element (FE) solver for the solution of 2D hydrodynamic equations (St-Venant equations), which has the capacity to manage Drying/Wetting conditions to track the evolution of the shoreline with respect to water level. MODELEUR can achieve all the phases of terrain data integration to produce a complete Numerical terrain model (TNM) based on a Finite Element mesh. In addition, MODELEUR facilitates the use and the control of hydrodynamic simulations.

The software is a highly interactive multi-window application, running on WindowsNT/95 PC platforms with Pentium processors. It is written in C++ with an internal architecture complying with the object-oriented programming philosophy. MODELEUR is made of different interconnected modules, such as Data management, display, construction of Numerical terrain model, simulation control, and various tools such as a data probe and a programmable calculator adapted to Finite Elements.

Infrastructure

To use the MODELEUR and HYDROSIM software, you must install them on a PC compatible computer and save them on the hard drive to have access to all the functions.

Hardware and software requirements

MODELEUR is a software designed to run on the WIN32 platforms, i.e. Windows 95 & 98, Windows NT 3.51 and 4.0. HYDROSIM is a WIN32 software in console mode, which can be run on the same platforms, although a UNIX version is also available.

The minimum configuration to install and run these software requires at least a Pentium 133 MHz computer with a minimum of 16 Megs of Random Access Memory (RAM) and 25 Megs of hard drive space.

Your screen must have a resolution of at least 800x600. A mouse is also essential to use the software.

These are strictly minimum requirements. Depending on the nature of your work, you may have to increase the capacity of certain components. For example, you will eventually need hard drive space to store your projects. If you increase the RAM capacity to 32 Megs, 64 MEGS or more, you will notice a better response time. A 15 or 17 inch screen with a resolution of 1024x748 will provide better comfort. Finally, the use of more powerful processors, such as Pentium II 333 MHz or 400 MHz, offers greater calculation capacities when simulating more complex cases.

The Windows NT operating system is a relatively more stable platform than Windows 95/98.

Installation

To install the software, access the installation medium and start the **setup.exe** program. The installer will be launched to guide you through the installation process of the software.

Physical installation on disk

When the installation is completed, you should find on your hard drive the installation directory which you have specified, for example **c:\Modeleur1.0a07**. In this directory, you will find several sub-directories to ensure the good functioning of the software.

Thus, you should see the following directories :

```
c:\Modeleur1.0a07
      aide
      bin
      doc
      etc
      filtres
      geodata
      langue
      projet
      simul
      terrain
```

The **aide** directory contains all the Help files essential to the comprehension of MODELEUR and HYDROSIM. Files with the **.hlp** extension may be consulted directly if wanted by simply double-clicking on them. The other files are used by the software to provide contextual help.

The **bin** directory contains all the execute files and the initialization files needed to use the software.

The **doc** directory contains the users guide and the documentation in PostScript format..

The **etc** directory contains several scripts and execute files that allow MODELEUR/HYDROSIM interactions.

The **filtres** directory contains the execute files used for data import and export operations.

The **geodata** directory contains the data needed for georeferences, including datum and predefined projections.

The **langue** directory contains the text and error messages translation files. The **.frc** extension contains the French version, the **.eng** extension contains the English version, and the **.esp** extension contains the Spanish version.

The **projet** directory contains a few demo projects to help you learn the software.

The **simul** directory will contain the simulations that you will launch with HYDROSIM.

The **terrain** directory contains the terrain data for the demo projects.

Installation in the programming environment

After the installation, you should find in the program bar the entry corresponding to Modeleur1.0a07. This entry should give you access to :

```
Modeleur1.0a07
     Modeleur
     Modeleur read me
     Hydrosim read me
     Modeleur registration
     Hydrosim registration
     Help
          General description
          Work procedure
          Project management
          Data display
          Controlling the display
          Using the tools
          Programming language
          Data model
          Data formats
          Data preparation
          Conducting a simulation
          Hydrosim 1.0a06
```

MODELEUR can be launched with the first entry of the menu and you can also consult the information on the new features of the current version of MODELEUR and HYDROSIM. You also have access to the documentation on the registration procedure for MODELEUR and HYDROSIM.

Finally, you have access to help concerning all the aspects of the use of MODELEUR and HYDROSIM. This is the on-line version of the written documentation. These help files can also be accessed through the MODELEUR interface depending on the context in which you will be when you press F1.

Launching

To launch MODELEUR, open the **Launch** menu, the **Programs** sub-item, the item **Modeleur1.0a07**. Select the item **Modeleur**. The execute file will launch MODELEUR after less than 20 seconds of processing.

The main MODELEUR window will appear. At this stage, you should be in a software **demo** version. You can verify that by opening the **About** box accessible in the **Help** menu, item **About...**.

To register MODELEUR and HYDROSIM, follow the registration procedure.

Registration procedure

To register the software, you need an access code associated with your installation. In fact, when you install the software, you are automatically in the **demo** version. This means that you can execute the software without any possibility of registration. However, you can browse through the software to take a look at their content and to assess the reach of these tools.

Write a e-mail describing what you want, providing the following information :

- Your name
- Name of the organization where you work
- The **info_licence.txt** file found in the \bin sub-directory of your installation.

Send your e-mail to the following address :

`Licence_Hydreau@inrs-eau.uquebec.ca`

Everything you need to register your software will be e-mailed to you. In fact, the software come in two modes :

1 - **Demo** licence: this is the default version at installation. Access to certain functions remains unauthorized. In general, you are unable to save your modifications.

2 - **Full** licence: this is the operational version, after registration. You will then have access to all the functions of the software. Duration of the license is either limited or unlimited.

You will get a section for each software which will have to be copied in the files **modeleur.ini** and **hydrosim.ini** found in the installation directory. In fact, they are found in the same directory as the execute files **modeleur.exe** and **hydrosim.exe**. Open these text files and copy the integral text at the end of the .ini files. Save.

Example of a section received :

```
[LICENCE_WBPO68BF]
VersionLogiciel=Modeleur 1.0a06 10 dec 1998 Dev
RegistrationKey=RK-DIkGoTd0-b0CtoYL0-zzE1gPtU-Lv9C0eoV
User=Jean Tremblay
Company=INRS-Eau
NbrConcurrentUsers=1
ExpirationDate=01/09/1997 - 23:59:59
NbrExecutions=-1
Modules=0xffffffff
```

Restart MODELEUR and you should find in the **About** box that the licence has changed from **demo** to **full**.

Description of the user interface

The MODELEUR/HYDROSIM software offers a graphic interface comprising the following elements :

- Main window
- Display windows
- Dialog boxes

Main window

The main window gives you access to all the functions of the software. This window contains several major elements discussed below.

First is the title bar at the top of the window. This bar contains the name of the software, in this case MODELEUR. The title is also completed by the name of the project currently loaded. On the left of the title bar is the system menu ; at the extreme right of the bar are the buttons to minimize, maximize and close the application.

Second, you will see the software menu bar. MODELEUR provides you with a series of menus with submenus to access the different functions.

Some of the items in the main menu will lead you directly to a dialog box or to a pop-up window. Other items on the main menu take you into specific modes ; you will then see one or several tool bars appear at the bottom of the main window.

In fact, all the items in the main menu followed by **[...]** indicate that you will be entering a specific mode. The function buttons appearing at the bottom of the main window are functions that are available **only** with these buttons.

Notes:

- The accesses to specific modes are mutually exclusive and you must leave a mode to access another one. The active mode is checked in front of the menu item.

Finally, the area between the menu at the top of the window and the function buttons at the bottom of the main window is known as the client space. The display windows will appear in this space.

The following tables describe the various menus and submenus.

- Project Menu
- Modeler Menu
- Simulation Menu
- Display Menu
- Tools Menu
- Window Menu
- Help Menu.

Project Menu

Menus	Options	Description
Project	*New...*	Displays the dialog box to create a new project. Specification of the directory, the source of data and the location in space.
	Open...	Displays the dialog box to open an existing project.
	Close	Close a project.
	Save...	Displays the dialog box to save the current project.
	Save As...	Displays the dialog box to save the project under a different name.
	Print...	Displays the dialog box to print the content of the active display window. Print on a printer, in a postscript file or a bitmap file.
	Import an entity...	Displays the dialog box to import an external entity into the current project. Source of the data, type of entity, import filter, and links to be established.
	Export an entity...	Displays the dialog box to export an entity from the current project. Selection of the type of data, choice of the entity, and export filter.
	Parameters...	Displays the dialog box to modify some parameters of the current project. Modification of the display area and of the location in

Menus	Options	Description
		space.
	Entity manager...	Displays the pop-up window to manage the entities of your project. You can rename, erase and consult the information.
	Exit	Leave the software.

Modeler Menu

Menus	Options	Description
Modeler	*Mesh [...]*	Takes you to the mesh generation mode. Gives access to the mesh partition conception tools, and access to mesh generation.
	Topography [...]	Takes you to the topography transfer on a mesh mode. Gives access to topography partition conception tools, specifies the sources of data, and transfers the topography on a selected mesh.
	Boundary conditions [...]	Takes you to the boundary conditions specification mode. Builds a boundary conditions partition from the geometry of an existing mesh partition. Gives access to the inflow and outflow conditions specification tools. Generates boundary conditions on a selected mesh.
	Wind [...]	Takes you to the wind characteristics specification mode. Gives access to the wind partition conception tools, specification of wind characteristics by surface, and generation of a wind Field on a selected mesh.
	Substrate [...]	Takes you to the substrate characteristics specification mode. Creation of a substrate partition from a grain size analysis, access to the substrate partition conception tools. Specification of substrates by surface and generation of a substrate Field on a selected mesh.
	Ice [...]	Takes you to the ice characteristics specification mode. Access to the ice partition conception tools. Specification of the ice characteristics and generation of an ice Field on a selected mesh.
	Friction [...]	Takes you to the other friction terms specification mode. Access to the friction partition conception tools. Specification of a friction coefficient and generation of a friction Field on a selected mesh.

Simulation Menu

Menus	Options	Description
Simulation	*Stationary...*	Displays the pop-up window to control the simulation. This window allows you to specify the simulation, to launch the simulation in interaction with the hydrodynamic simulator, and to recuperate the results.

Display Menu

Menus	Options	Description
Display	*Static [...]*	Gives access to the data static display mode. This mode provides the display of meshes, isolines, isosurfaces, vectors, nodes, and profiles.
	Cartographic dressing [...]	Gives access to the cartographic dressing mode. In this mode, you can add text, a scale bar, a North arrow, a grid, a legend.

Tools Menu

Menus	Options	Description
Tools	*Data probe...*	Displays the data probe pop-up window. You can view the position of the mouse in any coordinate system. You can view any scalar or vector field directly in the data probe.
	Function editor...	Displays the pop-up window of the function editor for the calculator. In this editor, you can conceive and validate functions usable by the calculator.
	Calculator...	Displays the calculator pop-up window. The calculator gives you access to a wide range of calculation generic possibilities. It is a specialized calculation tool for the scalar and vector fields of your project.
	Projection...	Displays the dialog box to project a field on another mesh. Allows to project data from a given mesh to another mesh.
	Profile [...]	Takes you to the profile conception mode. Gives access to the profile partition conception tools.

Window Menu

Menus	Options	Description

Menus	Options	Description
Window	*Create*	Create a display window.
	Close all	Close all the display windows.
	Cascade	Arrange the windows in cascade.
	Tile Horizontal	Arrange the windows in a horizontal tiling
	Tile Vertical	Arrange the windows in a vertical tiling
	Arrange icons	Arrange the icons of the minimized windows.

Help Menu

Menus	Options	Description
Help	*Content*	Access the table of contents of on-line help.
	Index	Access the index of on-line help.
	Search	Search help topics from a keyword.
	How to use help	Obtain help on how to use on-line help.
	About	View information on the software.

Display windows

In the software, the display windows are the site where information, generally spatial, is displayed. We can consider that these windows represent maps with overlying layers of data.

In fact, as soon as a window is created, a tool bar at the bottom of the main window allows you to manage this information. You can zoom in or zoom out, re-center the window on a selected point, return to a previous view... In addition, you can control the order of appearance of the various information layers in the window.

Besides the previous actions, the following basic actions can be executed on the display windows :

- Create a window
- Rearrange the windows
- Close all the windows

Create a window

To create a display window, select the option **Create** in the **Window** menu. The window appears with a default label.

Rearrange the windows

The display windows can be automatically rearranged in two ways :

- to arrange the windows in cascade, select the option **Cascade** in the **Window** menu.
- to arrange the windows in tiling, select the option **Tile Horizontal** or **Tile Vertical** in the **Window** menu.

Close all the windows

To close all the windows, select the option **Close All** in the **Window** menu.

Dialog boxes

When you use the software, you will come across various dialog boxes. They are either **modal** or **non modal**.

When a dialog box is **modal**, you must enter the information requested and confirm your choices to be able to continue.

When a dialog box is **non modal**, i.e. a pop-up window, you can put it aside temporarily and conduct other tasks. This type of window does not require that you complete the information immediately to continue working.

The boxes have a certain number of field types which allows consultation, entry and/or selection of information. Other elements of these boxes will launch a determined operation.

Following is the list of the various types of elements found in the dialog boxes, along with a description of their roles and uses :

1 - The **command buttons**. A command button such as OK, Cancel, Help will launch an operation. Simply clicking on such a button will launch the command.

2 - The **radio buttons**. In general, a radio button is used to give access to an associated configuration page. The radio button is always part of a group of buttons, and the choice of one of these buttons indicate an exclusive selection..

3 - The **check boxes**. In general, a check box allows you to select an option which may be active or not. By checking the box, you activate or deactivate the option specified.

4 - The **no-edit field**. The text which cannot be edited provides information for consultation only. It is also used to identify the other types of field.

5 - The **edit field**. In these fields, you can enter information to selected destinations. This type of field may be specialized to accept only numbers, if needed.

6 - The **drop-down list box**. In a drop-down list box, you can select an item among a list of items, without the possibility of modification.

7 - The **drop-down combo box**. In a drop-down combo box, you can select an item among a list of items. However, you can also modify the content.

8 - The **list box**. A list box displays a list of item to select from. Often, this type of list support multiple selection or double-click on an item. Sometimes, a pop-up menu is available upon right button click.

9 - The **tree view control**. The tree view control is a special list box that displays information as an indented outline based on hierarchical relationship. Often, functions are available by clicking or double-clicking, and a pop-up menu can be accessed by right button click. Finally, the tree view allows you to expand or collapse an outline by clicking on the + or the - respectively.

Chapter 2 Work procedure

The work procedure with the software MODELEUR/HYDROSIM consists of three main steps :

1 - Import terrain

2 - *Construct the numerical terrain model*

3 - *Conduct a simulation*

This work procedure is only indicative and is, by nature, iterative. You will necessarily have to repeat this procedure several times, and even create several branches to test various alternatives.

Import terrain data

The importation of terrain data allows you to integrate external data to construct the numerical terrain model. To model a river, you must obtain the most precise description possible.

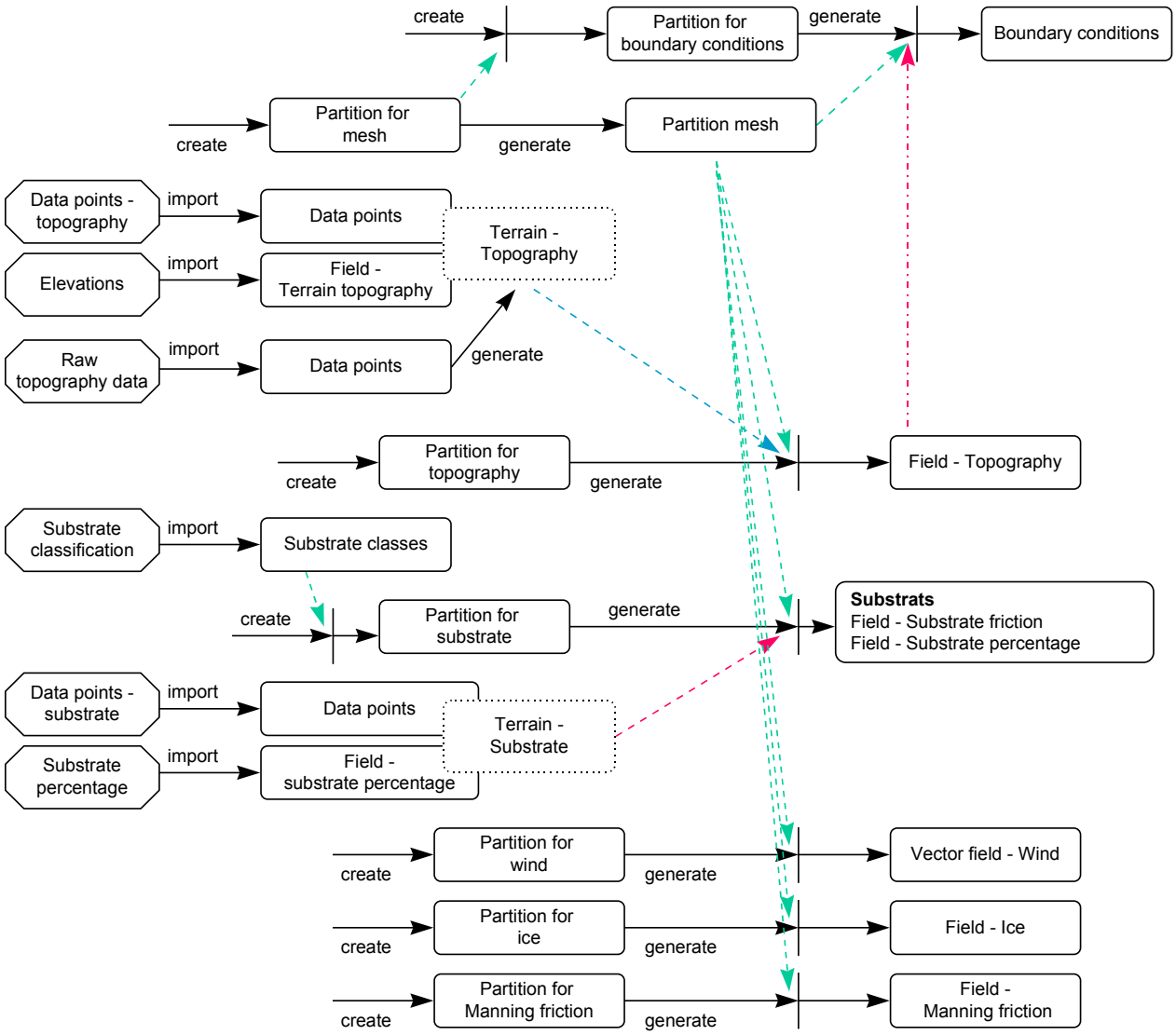
First, you need a description of the topography, which may combine several sources. To import each source of topography data, you must structure the information in compliance with the data format. (See the documentation on data formatAID_FORMATS_DONNEES@formdonn.hlp.) In the case of topography, you need **data points** and a **scalar field - topography** which you will import in that order.

Other terrain data may be used. For example, you may import data on **substrate classification** as surveyed in the field, as well as detailed substrate analyses.

(See the documentation on data preparationAID_PREPARATION_DONNEES@prepdonn.hlp.)

Construct the numerical terrain model

Once you have completed the importation into your project of all the data available, you can start building the numerical terrain model. MODELEUR provides a series of functions to guide you through this process.



You should normally start by building a simulation mesh. This mesh will hold as many nodes as you estimate necessary. The **mesh partition** mode allows you to specify the geometry of the mesh partition, as well as the level of refinement of the mesh. The end result is a **mesh** for hydrodynamic simulation.

The second important step is to transfer the topography on the mesh. The **topography partition** mode is used for this task. Using the topography partition, you can specify the origin of the topography data by zone. The goal is to obtain a global **scalar field - topography** covering the entire domain specified by the mesh ; you must be able to provide topography data for the entire simulation mesh. Topography can then be transferred on the hydrodynamics mesh.

The third important step is to specify the boundary conditions of your domain. The **boundary conditions partition** mode allows you to specify the inflows and the outflows in your domain. You

can create and specify several boundary conditions to simulate floods, low water, and other characteristic discharges. The end results are **boundary conditions** associated to a hydrodynamics mesh.

The other steps are more optional.

For example, you can modelize the nature of the river bed in order to estimate the level of friction associated with morphology. Also, the nature of the bottom can be very important in certain types of environmental studies such as fish habitat quality assessments. Therefore, you have access to the **substrate partition** mode which allows to characterize the substrates on the basis of a classification of substrates and the percentage observed in the domain. The mode produces two results : the **scalar field - substrate friction** and the **field - percentage of substrate** associated to a hydrodynamics mesh.

There is also a **wind partition** mode to modelize dominant winds by zone. The end result is a **vector field - wind**.

There is also an **ice partition** mode to modelize ice cover. The end result is a **field - ice**.

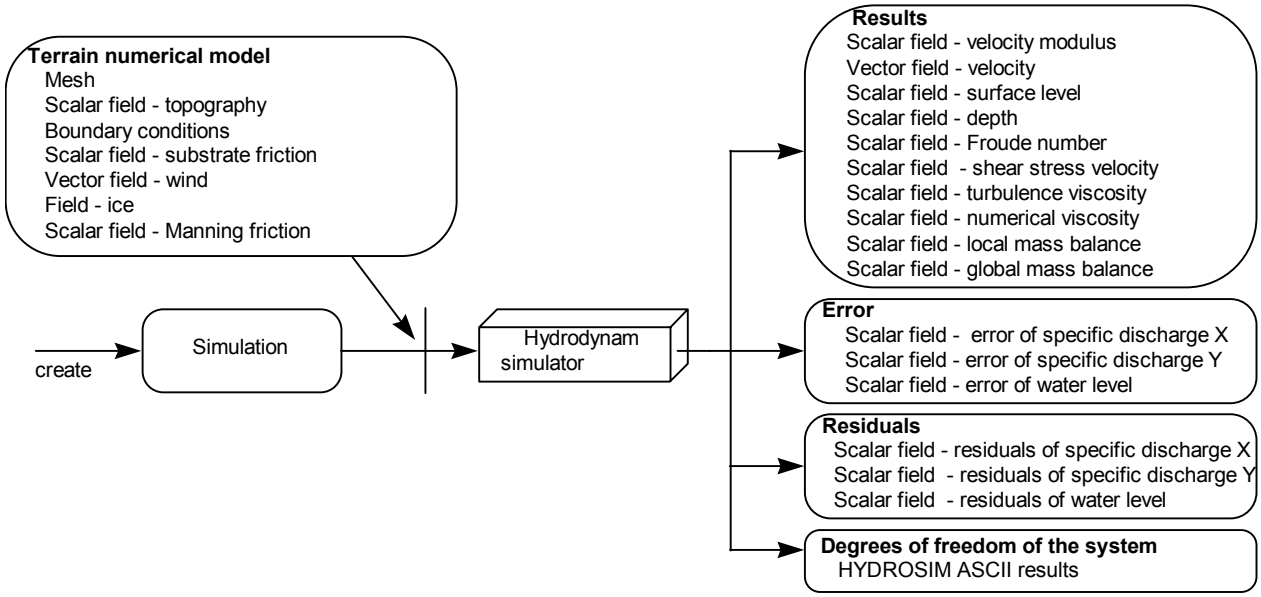
Finally, there is a **Manning friction partition** mode to specify Manning coefficients directly by zone. The end result is a **scalar field - Manning friction**.

For more information, read the documentation on data preparationAID_PREPARATION_DONNEES@prepdonn.hlp.

Conduct a simulation

Once your numerical terrain model is completed, the software MODELEUR/HYDROSIM allows you to conduct hydrodynamics simulations.

In MODELEUR, you have access to a simulation conception console from which you integrate everything that is needed to specify a simulation. You can then launch the simulation and recuperate the results generated by the hydrodynamics simulator.


Terrain numerical model
Mesh
Scalar field - topography
Boundary conditions
Scalar field - substrate friction
Vector field - wind
Field - ice
Scalar field - Manning friction
create
Simulation
Hydrodynam simulator
Results
Scalar field - velocity modulus
Vector field - velocity
Scalar field - surface level
Scalar field - depth
Scalar field - Froude number
Scalar field - shear stress velocity
Scalar field - turbulence viscosity
Scalar field - numerical viscosity
Scalar field - local mass balance
Scalar field - global mass balance
Error
Scalar field - error of specific discharge X
Scalar field - error of specific discharge Y
Scalar field - error of water level
Residuals
Scalar field - residuals of specific discharge X
Scalar field - residuals of specific discharge Y
Scalar field - residuals of water level
Degrees of freedom of the system
HYDROSIM ASCII results

Essentially, the input data are :

- The hydrodynamics **mesh**
- The **scalar field - topography** related to the mesh
- Friction coefficients from the **scalar field - substrate friction**, the **scalar field - Manning friction**, the **field - ice** all related to the mesh. These friction sources will be cumulated in a Manning coefficient.
- The selected **boundary conditions** related to the mesh.
- The **vector field** related to the mesh.

In addition to these data taken from the numerical terrain model, you will have to specify :

- The global properties
- The initial conditions
- The resolution scheme
- The results to recover

For more information on the conception of the hydrodynamics simulation, see the documentation on launching a simulationAID_SIMULATION_STATIONNAIRE@realsimu.hlp and the documentation on the hydrodynamics simulator.

The hydrodynamics simulator can then be asked to calculate the results of the simulation.

The simulator will produce the following results :

- The residuals:
  - scalar field - residuals of specific discharge in X

- scalar field - residuals of specific discharge in Y
- scalar field - residuals of water level

- The errors:
  - scalar field - error specific discharge in X
  - scalar field - error specific discharge in Y
  - scalar field - error of water level
- The results:
  - scalar field - velocity modulus
  - vector field - velocity
  - scalar field - surface level
  - scalar field - depth
  - scalar field - Froude number
  - scalar field - shear stress velocity
  - scalar field - physical viscosity
  - scalar field - numerical viscosity
  - scalar field - local mass conservation
  - scalar field - global mass conservation
  - HYDROSIM initial data :
  - HYDROSIM ASCII results

For more information on the nature and the significance of these results, see the documentation on the hydrodynamic simulator.

Chapter 3 Project management

Developing a numerical terrain model, executing numerical simulations, analyzing and interpreting results imply a large amount of interrelated data. To handle all these data while keeping them cohesive is a complex but necessary task. The software releases you from these constraints by offering a more natural view of the data in the form of projects grouping all the data on the study domain. Data are presented in the form of entities and of interrelations. For more details regarding the data model of the software, consult the appendix on the **data model**.

The possible actions on projects are :

- Create a new project
- Open an existing project
- Save the current project
- Save As...to save a project under a different name
- Close the project
- Modify the parameters of the current project.

The possible actions on the project's entities are :

- Import an entity
- Export an entity
- Handling the project's entities.

Create a new project

When creating a new project, a certain number of information must be provided. You must specify the directory in which the project will be installed, the projects on which work can be based (optional) and set the parameters for spatial localization.

To create a new project, follow these steps :

1 - Select the option **New...** in the **Project** menu. The dialog box **New project** will appear.

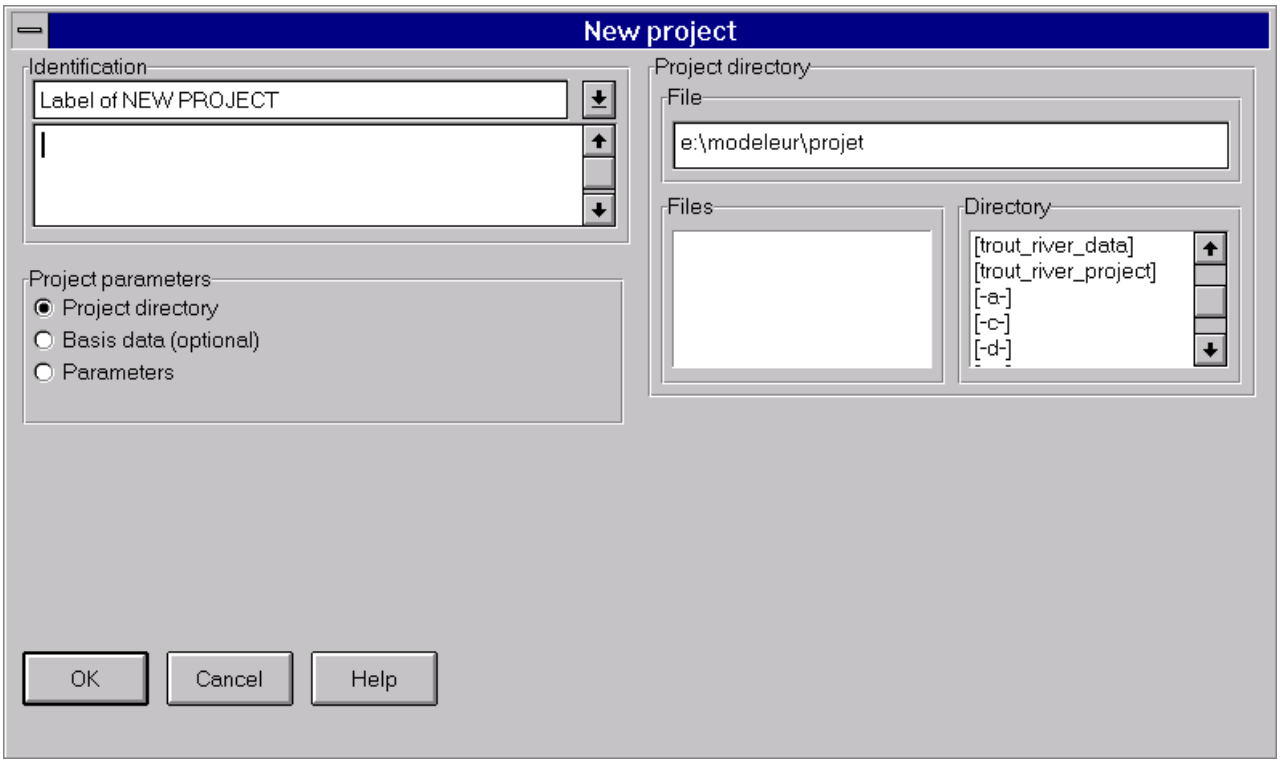
2 - Identify the new project in the section **Identification**. (See *Identify a new project*)

3 - Specify the directory for the new project in the **Project directory** page. (See *Specify the directory for the project*)

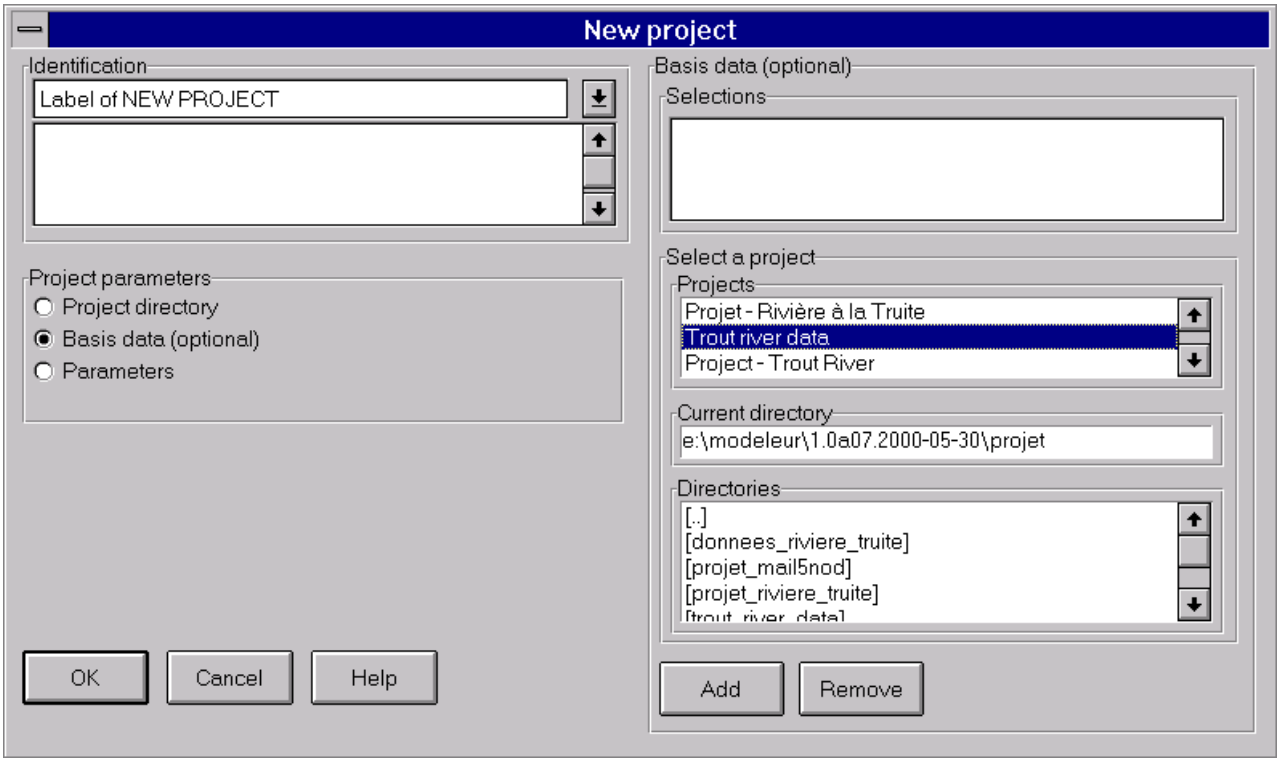
4 - If wanted, select in the **Basis data** page, the existing project(s) on which to base the new project. (See *Base a project on existing data*)

5 - Specify the positioning parameters and the display area in the **Project parameters** page. (See *Georeference and positioning*)

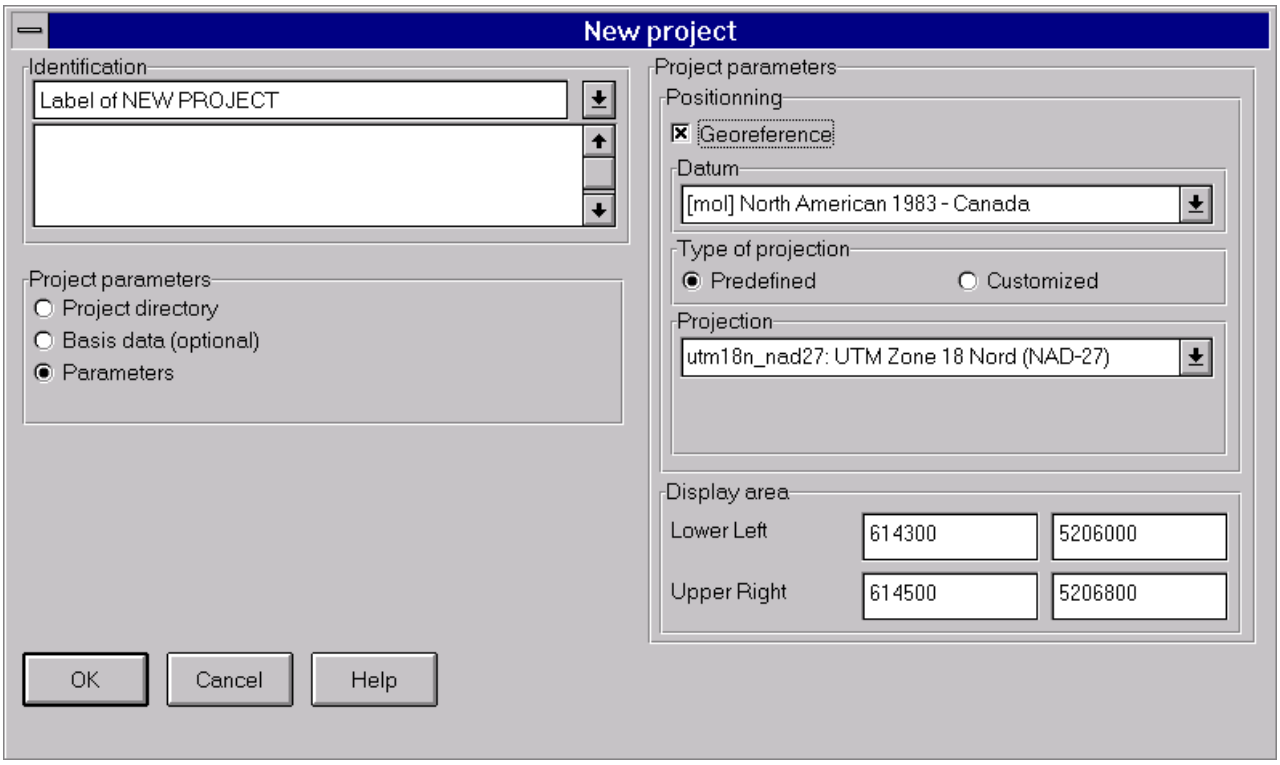
6 - Click on **OK**. After validating the parameters, the software creates the new project which becomes the current project. If a project was already loaded, it will be closed first.



Example -1 New project dialog box **1/3**



Example -2 New project dialog box **2/3**



Example -3 New project dialog box **3/3**

Identify a new project

You can label and give a description of your project in the section **Identification** of the **New project** dialog box :

1 - Enter the label of the project in the upper part of the section.

2 - Type a description of the project in the lower part of the section.

Specify the directory for the project

When creating a new project, you must specify the directory in which the data will be saved :

1 - Activate the option **Directory** in the section **Project's parameters** in the **New project** dialog box**.** The **Project directory** page will appear in the right end portion of the dialog box.

2 - By double-clicking, select the pathway to the new project directory in the section **Directory.** As you make your selections, the access pathway is displayed in the section **File**.

3 - In the section **File,** complete the access pathway by giving a name to the directory.

Note:

- For the software to create the new directory, the following conditions must be satisfied :
  I. the directory cannot already exist ;
  II. the parent directory must exist and be accessible in read/write format.

Base a project on existing data

When creating a new project, it can be useful to load data from one or many existing projects for further modeling.

In the section **Project's parameters** of the ***New project*** dialog box, the option **Basis data** gives you access to the page where you can add existing projects from which to create the new project This feature of the software allows to build a new project from existing data useful for your analysis.

Follow these steps :

1 - Double-click on the directory of the basis project in the section **Directory.** If the selected directory contains valid projects, they will be displayed in the section **Projects.** As you make your selections, the current access pathway is displayed in the section **Current directory**.

2 - Select a project to be added and click on **Add**. The label of the project will be added to the list in the section **Selections**. To delete an entry in this list, select it and click on **Remove**.

Notes :

- If you select several projects, the software verifies if they are compatible in terms of georeference. If they are not, an error message informs you of the situation and the selection is ignored.
- If you add a project already included, a message informs you of the situation and the selection is ignored.
- In the case of a valid selection, the edit fields of the groups **Display area** and **Positioning** in the **Project's parameters** page are automatically updated according to the data contained in the basis project. (See *Georeference and positioning*).

Georeference and positioning

When creating a new project, you must provide information for the software to properly locate the data in space. First, you must specify whether your project will be georeferenced or not. If you do not want your project to be georeferenced, the software will generate a cartesian coordinate system which will not transform the spatial data. If you want your project to be georeferenced, you must select the necessary parameters to specify an internal coordinate system for your project. In fact, you have to specify a datum and a projection. This internal coordinate system must necessarily be a cartographic coordinate system. (For more information, see the documentation on **proj**.)

You must also define a display area, i. e. the rectangular portion of your project with which you want to work. The display area must be specified in the coordinate system selected.

To specify these parameters, follow these steps :

1 - Activate the option **Parameters** in the section **Project's parameters** in the **New project** dialog box**.** The **Project's parameters** page will appear in the right hand portion of the dialog box.

2 - If you want your project to be georeferenced, check-off the **georeference** option**.** You must then select a datum in the

list of predefined datum and a projection in the list of predefined projections. It is also possible to provide a customized projection.

3 - In the first and second parts of the field **Bottom left** in the section **Display area,** enter the respective x and y coordinates of the lower left corner of the area.

4 - In the first and second parts of the field **Top right,** enter the respective x and y coordinates of the upper right hand corner of the area.

Notes:

- When the new project is based on an existing project (see :*Base a project on existing data*), its positioning and display area parameters are automatically initialized with the values of the basis project.
- The display area must form a non-degenerated rectangle where the lower left corner has x and y coordinates inferior to the upper right hand corner.
- The display area must necessarily be specified in the coordinate system selected.
  I. For example, if you use the **North American 1983 - Canada** datum and the **UTM Zone 18 North** projection, your display area must be specified in UTM (NAD83) coordinates.
  II. If you used a non-georeferenced system, your display area must be specified in the same coordinate system as your data ; the software does not transform coordinates.
- The customized projections allow to create a projection adapted to your specific requirements. Thus, if your project must cover several UTM zones, you must select a projection to represent several zones at once, with the least distortion possible. (See the documentation on **proj**)

Open an existing project

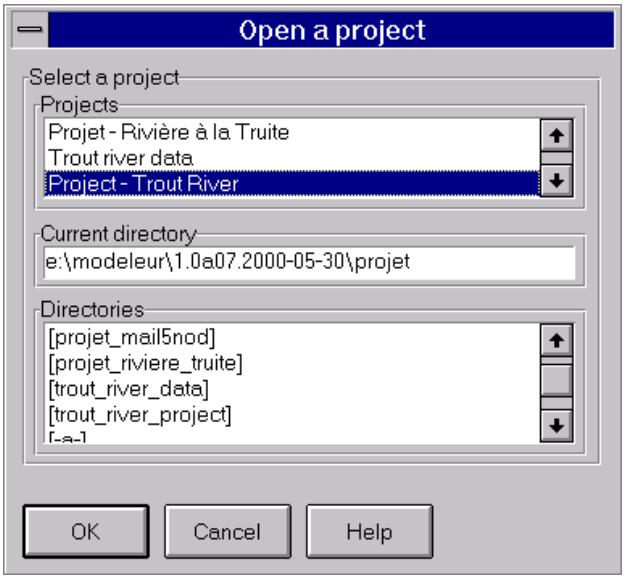
Other than consulting the on-line help or creating a new project, no operation can be effected if a project has not been formerly opened. To open a project, follow these steps :

1 - Select the option **Open...** in the **Project** menu. The **Open a project** dialog box will appear.

2 - In the section **Directories**, search the disks and directories for existing projects by double-clicking**.** If project directories are found in the current directory, they will be displayed in the section **Projects.** As you make your selection, the current access pathway is displayed in the section **Current directory**.

3 - In the section **Projects,** select the project to open.

4 - Click on **OK**. If a project is already opened, it will be closed before the selected project is opened.



Example -4 Open a project dialog box

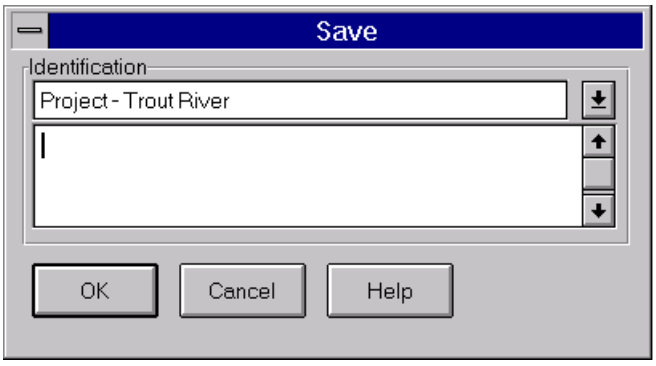
Save the current project

While working with a project, you modify and add entities in the database. These modifications and additions are conducted in the system memory and could therefore be lost in case of a program crash or a hardware problem. You can save at will the whole project in its current state :

1 - Select the option **Save...** in the **Project** menu. The **Save** dialog box will appear.

2 - If you wish, you can modify the label and the description of the project.

3 - Click on **OK**.



Example -5 Save dialog box

Save As...to save a project under a different name

With the option Save As, it is possible to save a project under a different name. This procedure saves the whole project in a new directory in its current state.

To Save As, follow these steps :

1 - Select the option **Save As...** in the **Project** menu. The **Save As** dialog box will appear.

2 - Enter the label of the project in the upper part of the group **Identification**.

3 - If you wish, you can type a description of the project in the lower part of the group **Identification**.

4 - In the section **Directory** of the group **Directory selection**, double-click on the access pathway of the new project directory. As you make your selections, the current access pathway is displayed in the section **File**.

5 - In the section **File,** complete the access pathway by naming the directory as desired.

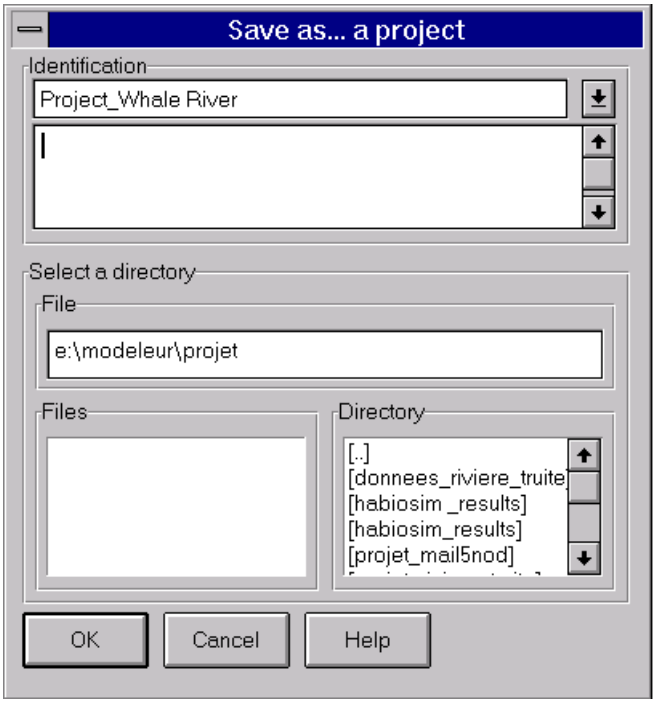
6 - Click on **OK**. After validation of the entries, the specified directory is created and the files of the current project are copied. The new project becomes the current project.

Note:

- For the software to create the new directory, the following conditions must be satisfied :

  I. the directory cannot already exist ;

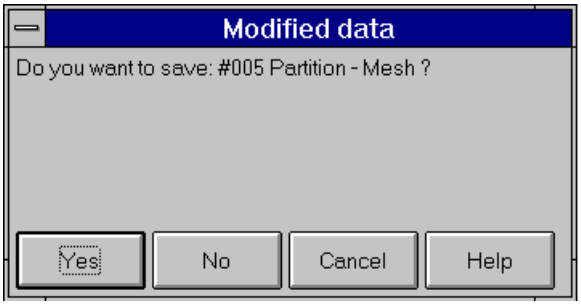
  II. the parent directory must exist and be accessible in read/write format.



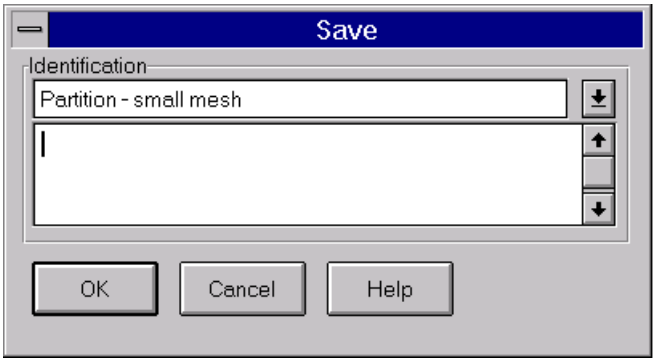
Example -6 Save As dialog box

Close the project

When a project is ready to be closed, the software verifies that all data have been saved. The software will ask you as often as needed if you wish to save a given data. You can at anytime cancel the closure by clicking on **Cancel**. If you do not want to save a given data, click on **No**. If you wish to save it, click on **Yes**.



If it is a new data without a label and a description, you will be asked to provide them. Click on **Ok** to confirm or on **Cancel** to cancel.



Closure can be triggered in various ways. If you want to create a new project, open another project, or leave the software while there is a project opened, closure of the current project will occur.

Modify the parameters of the current project

When creating the project, you specified the positioning and display area parameters (see *Georeference and positioning*).

You can modify these parameters at will. However, you cannot switch from a georeferenced system to a non-georeferenced system, or vice-versa.

It is highly advisable to close all display windows before doing this operation, especially if you are planning to modify the projection used. In this case, there will be a discrepancy between what is already displayed in a window and what will be displayed after.

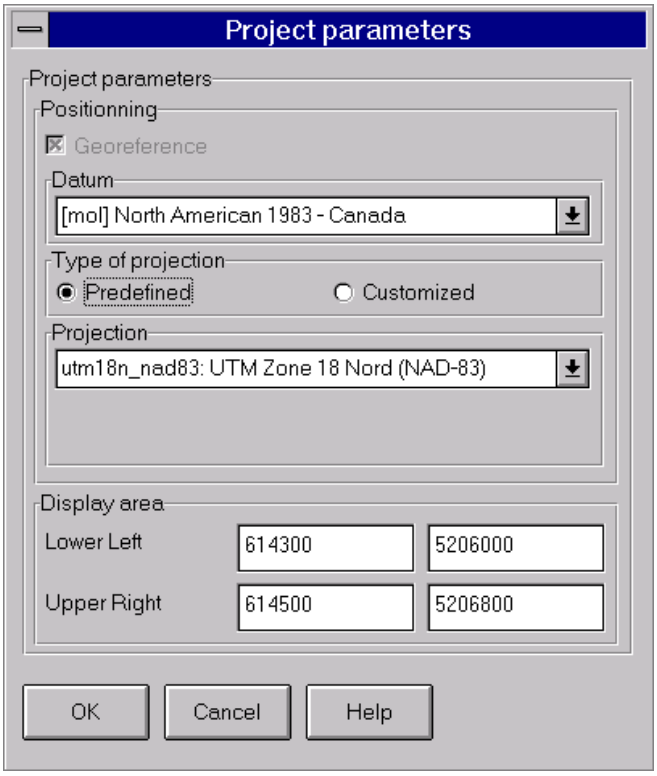
1 - Select the option **Parameters...** in the **Project** menu. The **Project's parameters** dialog box will appear.

2 - You can modify any parameter, but you must keep in mind that the display area must be specified in the coordinate system selected.

3 - Click on **OK ;** validations will be conducted before the operation is applied.

Notes :

- If you modify the projection, a new coordinate system will be generated in the database to represent the new internal coordinate system. However, the old coordinate system will remain available in the database to protect data which may be linked to it.



Example -7 Project's parameters dialog box

Import an entity

In modeling work, it is fundamental to be able to import data in a project, i. e. to integrate external data. Modeling is often based on data from different sources : data from field surveys, cartographic data, simulation results, etc. For these data to be used in a project, they must be incorporated in compliance with the existing data model.

The import function of the database allows you to integrate external data in your project in a controlled environment to maintain cohesion of the project database. It is based on the notion of file *formats* since each type of data imposes a precise format for importation in the database. (See the documentation on **data formats**). Also the imported data must comply with a precise data model. In fact, some data must be linked to existing entities in the database when imported. (See the documentation on **data model**).

The formats imposed on the data can be modified through the use of importation filters. For most of the data to import, the database can use a special filter which does not modify the format before importation, i.e. a neutral filter. However, if the data format required is not convenient for your purposes, you can write a special program, i.e. a filter, to pre-process your external data prior to the actual importation into the database. (See the documentation on **import/export filters**.)

Depending on the type of entity to be imported, the procedure may have specific features, but the basic steps always remain the following :

1 - Select the option **Import Entity...** in the **Project** menu. The **Import an entity** dialog box will appear.

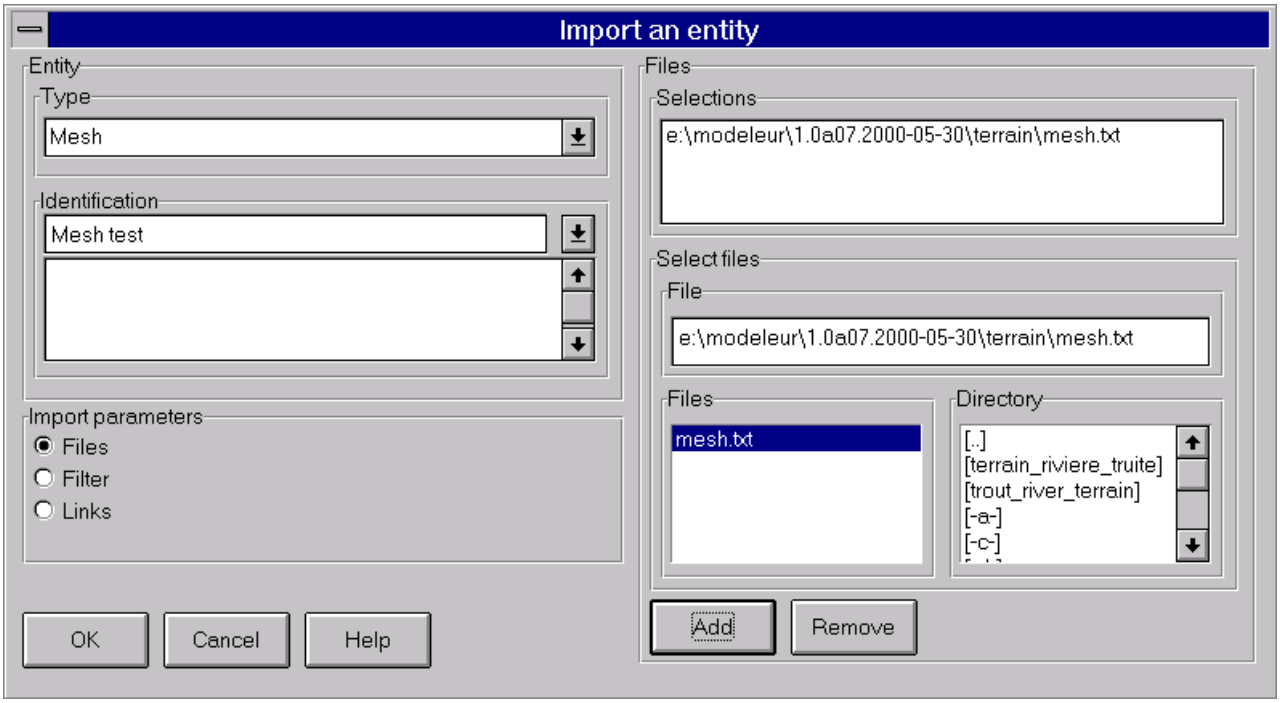
2 - Identify the entity to be imported in the section **Entity.** (See *Identify an entity to be imported*)

3 - In the **Files** page, select the file(s) containing the data of the entity to be imported. (See *Select the files to be imported*)

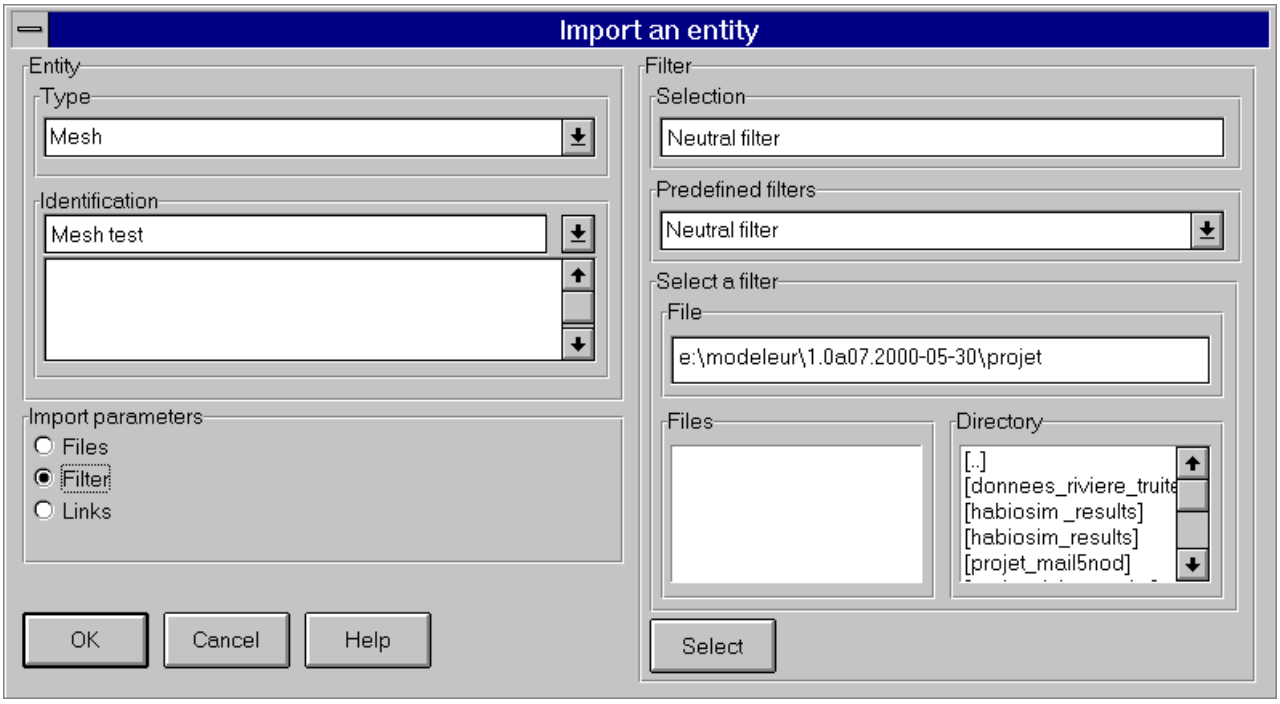
4 - In the **Filter** page**,** select the filter to be used to transcribe the data into the intermediate format. (See *Select a filter*)

5 - In the **Links** page, select the project entities to which the imported entity will be linked**.** (See *Select the links for an entity to be imported*)

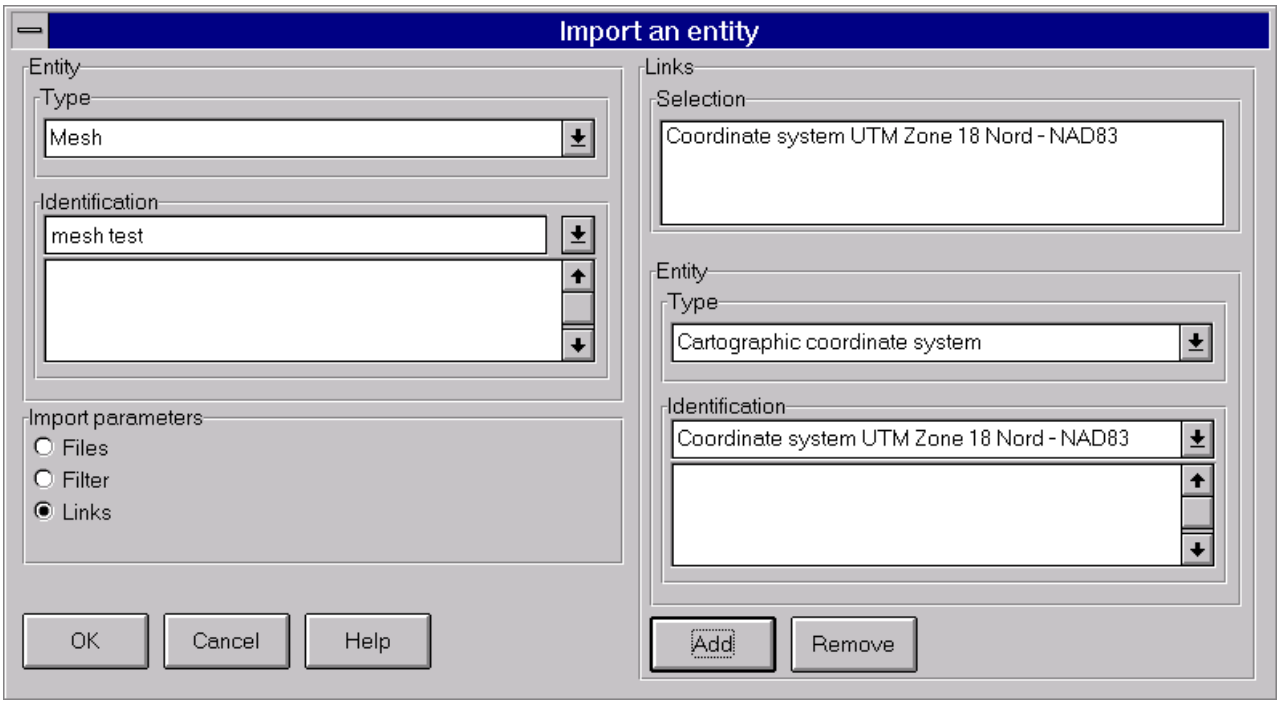
6 - Click on **OK**. The database will first validate the parameters, and then import the data which becomes a new entity available in the project.



Example -8 Import an entity dialog box **1/3**



Example -9 Import an entity dialog box **2/3**



Example -10 Import an entity dialog box **3/3**

Identify an entity to be imported

The section **Entity** in the **Import an entity** dialog box allows you to specify the type of entity to be imported. You must also provide a label and a description :

1 - In the section **Type**, select the type of entity. The list contains all the types available in the database for import/export.

2 - In the upper part of the section **Identification**, enter the desired label for the entity. You may draw from the labels of entities of the same type present in the project by consulting the drop-down list associated with this section. However, the name of the entity must be unique among the entities of the same type.

3 - In the lower part of the section **Identification,** enter a description of the entity.

Select the files to be imported

Before importing an entity in a project, the software must first know in which external file(s) the data are located. To select the file(s), follow these steps :

1 - Activate the option **Files** in the section **Import Parameters** of the **Import an entity** dialog box**.** The **Files** page will appear in the right part of the dialog box.

2 - Double-click on the wanted directory in the section **Directory.** The files contained in this directory will be displayed in the section **Files.** As you make your selections, the current access pathway is displayed in the section **File**.

3 - Double-click on the desired file in the section **Files**. The access pathway and the file name are displayed in the section **File**.

4 - Click on the button **Add**. The access pathway and the file name are added to the list in the section **Selections**. To remove an entry from this list, select it and click on the button **Remove**.

Select a filter

To convert the data of the entity to be imported into an intermediate format recognizable by the database, you must

select the filter to use for the conversion. If you provide data compatible with the suggested intermediate format, select the neutral filter.

1 - Activate the option **Filter** in the section **Import Parameters** in the **Import an entity** dialog box**.** The **Filter** page will appear in the right part of the dialog box.

2 - If you want one of the predefined filters, select it in the drop-down list of the section **Predefined filters.** The selected filter is added in the section **Selection.** If you want another filter, proceed with steps 3, 4 and 5.

3 - Double-click on the wanted directory in the section **Directory.** The program files contained in the directory are displayed in the section **Files.** As you make your selections, the current access pathway is displayed in the section **File**.

4 - Double-click on the desired file in the section **Files**. The access pathway and the file name are displayed in the section **File**.

5 - Click on the button **Select**. The access pathway and the file name are displayed in the section **Selection**.

Select the links for an entity to be imported

Certain types of entities cannot exist individually without going against the data model of the database. For example, any field must be linked to a mesh. To import such an entity, you must specify the existing entities to which it must be linked :

1 - Activate the option **Links** in the section **Import Parameters** of the **Import an entity** dialog box**.** The **Links** page will appear in the right part of the dialog box.

2 - In the drop-down list of the section **Type**, select the type of entity to which the imported entity must be linked. The list of all the project entities corresponding to the selected type becomes available in the upper part of the section **Identification**.

3 - Select the entity to which the imported entity will be linked in the drop-down list of the section **Identification**. Its description will appear in the lower part of the section.

4 - Click on the button **Add**. The label of the entity selected is added to the list of the section **Selection**. To remove an entry from this list, select it and click on the button **Remove**.

Notes :

- To have a complete description of all the links needed, consult the documentation on the **data model**.

Export an entity

The database can export entities present in the current project toward external destinations. Thus, data can be shared between different projects and different systems.

To export an entity, follow these steps :

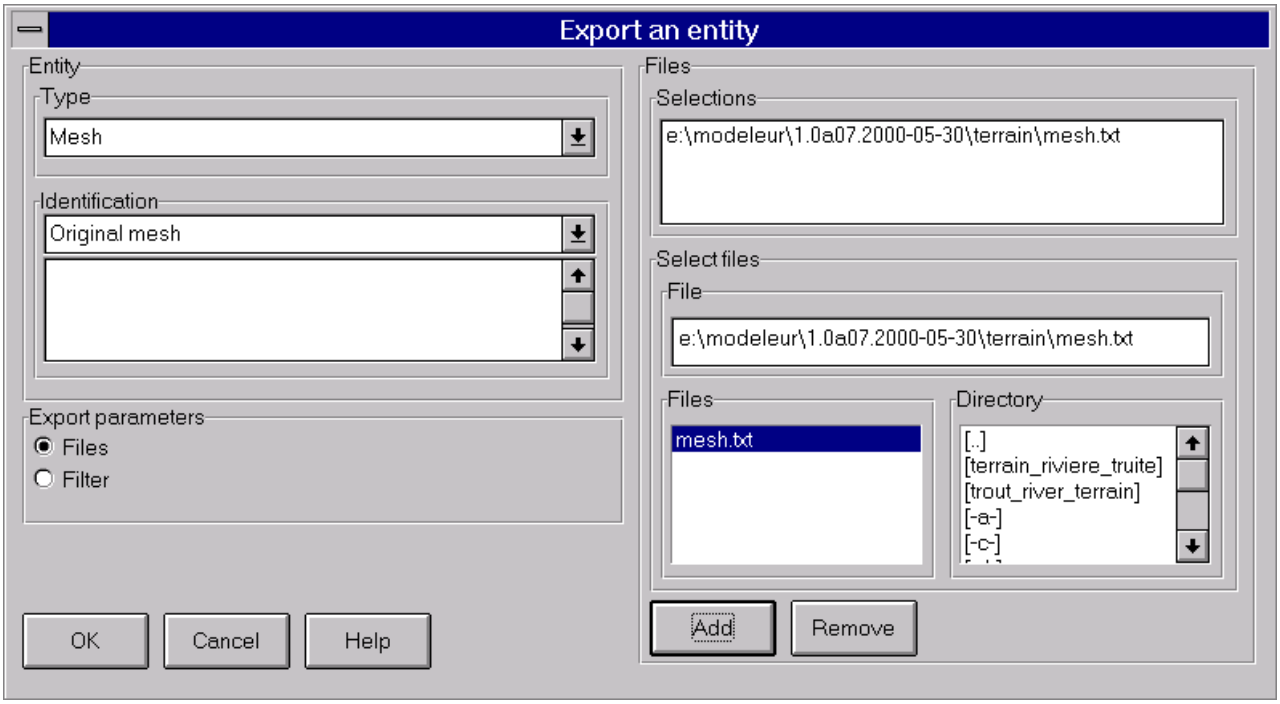
1 - Select the option **Export Entity...** in the **Project** menu. The **Export an entity** dialog box will appear.

2 - Identify the entity to be exported in the section **Entity.** (See *Identify an entity to be exported*).

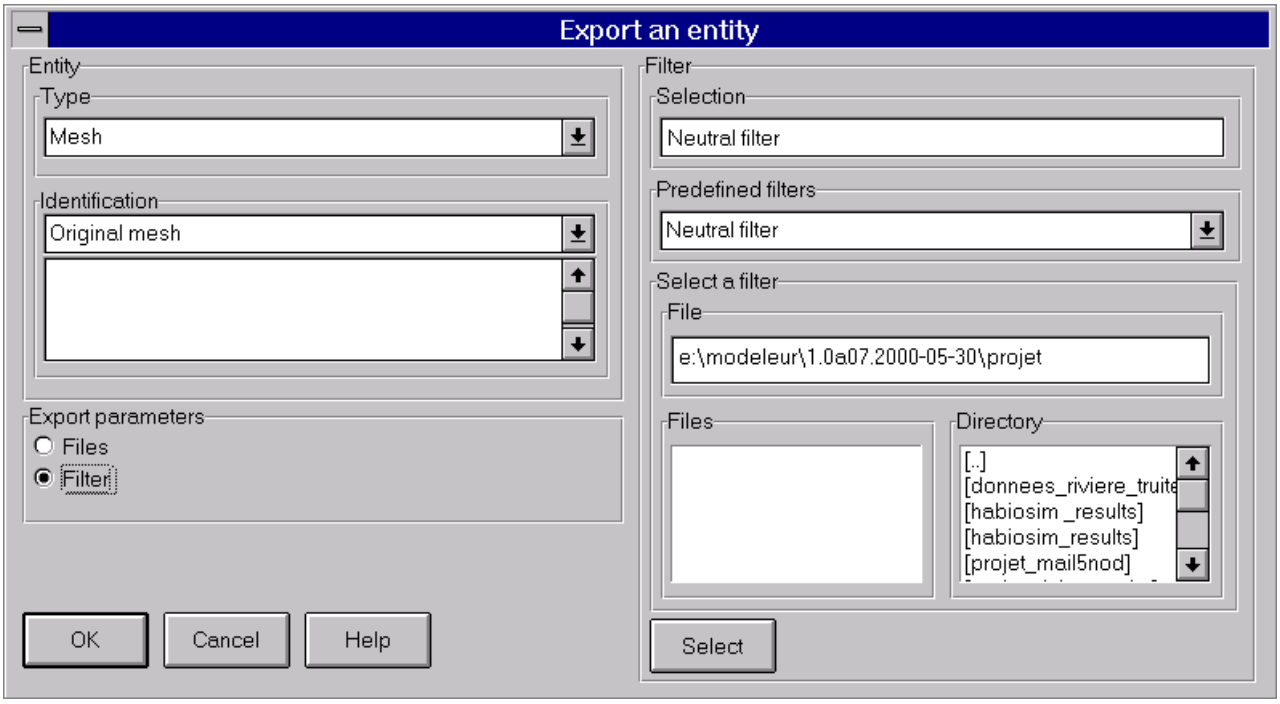
3 - In the **Files** page, specify the directory and the name of the file(S) in which will be written the data of the entity to be exported by the filter. (See *Specify the file(s) to be* exported).

4 - In the **Filter** page**,** select the filter needed to transcribe the data in the external format. (See *Select a filter*).

5 - Click on **OK**. After validating the parameters, the database exports the entity.



Example -11 Export an entity dialog box **1/2**



Example -12 Export an entity dialog box **2/2**

Identify an entity to be exported

The section **Entity** of the **Export an entity** dialog box allows you to select the entity to be exported :

1 - In the section **Type**, select the type of the entity. All the entities of this type present in the project become available in the drop-down list of the section **Identification**.

2 - In the drop-down list of the section **Identification,** select the entity you wish to export. Its description, if there is one, is displayed in the lower part of the section.

Specify the file(s) to be exported

1 - Activate the option **Files** in the section **Export Parameters** of the **Export an entity** dialog box**.** The **Files** page will appear in the right part of the dialog box.

2 - In the section **Directory,** double-click on the directory where the export file should be moved to. The files in this directory are then displayed in the section **Files.** As you make your

selections, the current access pathway is displayed in the section **File**.

3 - Enter the name you wish to give to the export file in the section **File**. You can also select an existing file in the section **Files.** In this case, the software will ask you to confirm the replacement of the file.

4 - Click on the button **Add**. The access pathway and the file name are added to the list of the section **Selections**. To remove an entry from this list, select it and click on the button **Remove**.

Note:

- For the software to create the export file(s), the parent directory must exist and be accessible in read/write format.

Select a filter

1 - Select the option **Filter** in the section **Export Parameters** of the **Export an entity** dialog box**.** The **Filter** page will appear in the right part of the dialog box.

2 - If you want one of the predefined filters, select it in the drop-down list of the section **Predefined filters.** The selected filter is added in the section **Selection.** If you want another filter, proceed with steps 3, 4 and 5.

3 - Double-click on the wanted directory in the section **Directory.** The program files contained in the directory are displayed in the section **Files.** As you make your selections, the current access pathway is displayed in the section **File**.

4 - Double-click on the desired file in the section **Files**. The access pathway and the file name are displayed in the section **File**.

5 - Click on the button **Select**. The access pathway and the file name are displayed in the section **Selection**.

Handling the project's entities

Modeling operations often imply the creation (or the importation) of a large number of interrelated entities. Therefore, it rapidly becomes arduous to have a precise overview of the project. Also, modeling being iterative by nature, entities often become unnecessary.

The software provides you with the possibility to view from different angles the entities of the project, to see their characteristics and the links between entities. It is also possible

to delete entities, or to modify their label and their description in a controlled environment protecting the integrity of the database.

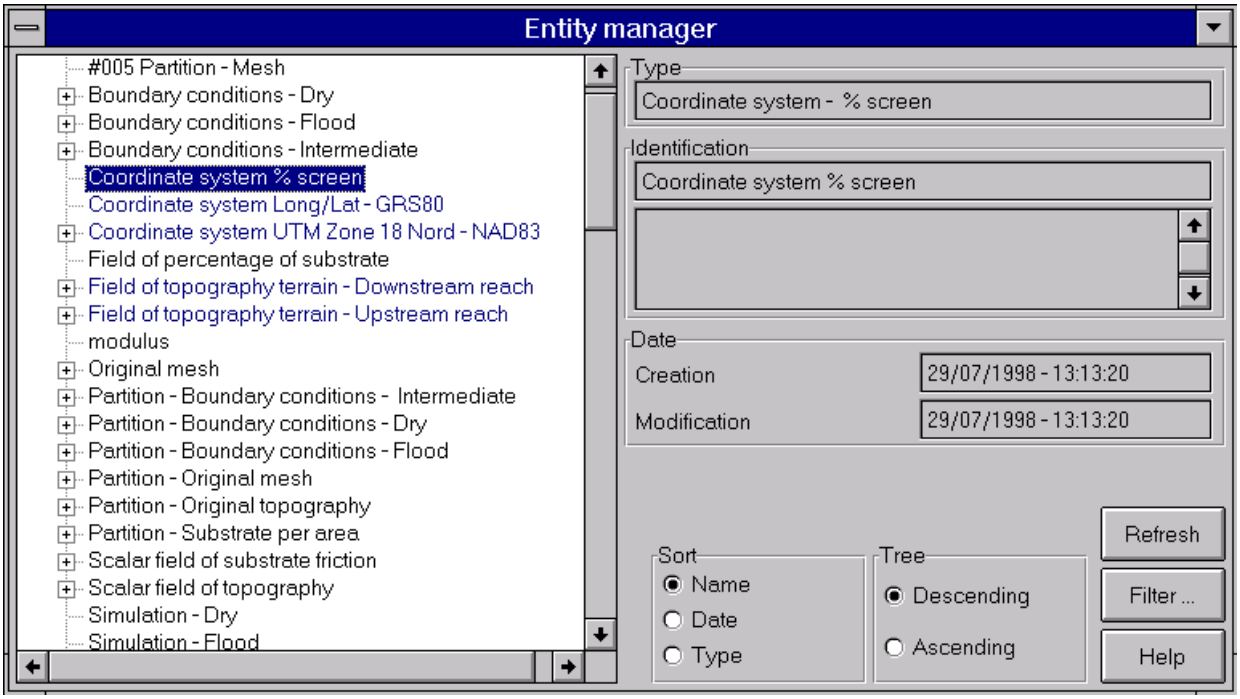
However, the projects included in the current project contain data which cannot be modified or destroyed by the user.

To access these functions, select the option **Entity manager...** in the **Project** menu. The pop-up window **Entity manager** will appear on the screen. The left part of the window shows a tree view control list of the project's entities. The content of this list depends on the parameters specified in the lower right portion of the window. The upper right portion gives some information relating to the entity selected in the list.

A colour code is used to highlight the origin of the entities. Entities shown in black are the entities of the current project, i.e. those which can be modified or deleted. Each other colour represents the origin of a different project.

Through the entity manager, you can :

- Display a sorted list of the entities
- Display a list of the entities on selected criteria
- View the links between entities
- Modify the label and/or the description of an entity
- Delete an entity from the project
- Display information on the characteristics of the entity



Example -13 *Entity manager* pop-up window

Display a sorted list of the entities

The entities displayed in the list of the **Entity manager** window are sorted according to the criteria indicated by the option selected in the group **Sort**. To change this sorting criteria, select the appropriate option. Thus, entities can be sorted :

- by ascending alphabetical order of the **name**
- by order of the **date** of the last modification (from the most recent to the oldest)
- by order of the **type** of entity.

Display a list of the entities on selected criteria

Whether to work with a specific group of entities or to reduce the size of the list to a more practical dimension, it may be useful to obtain a selective list of the project's entities. Follow these steps :

1 - In the **Entity manager** window, click on the button **Filter**. The ***Filter*** dialog box will appear.

2 - In the list of the group **Request**, select the characteristic on which you want to establish a criteria. The name of the characteristic selected will appear in the first field of the group **Criteria,** followed by the operator and the value forming the criteria currently applied to this characteristic.

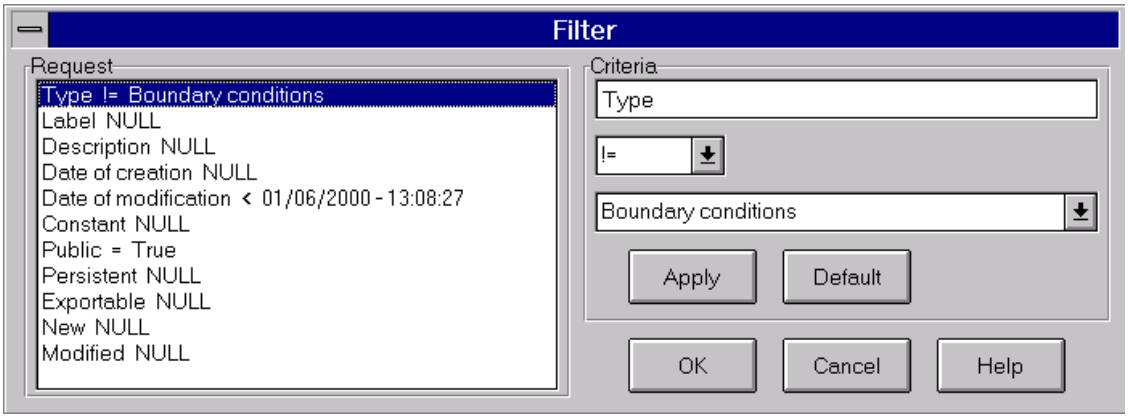
3 - Select the desired operator in the drop-down list of the group **Request**. The NIL operator is used to indicate that a criteria should not be applied on the characteristic selected.

4 - Depending on the characteristic involved, enter or select the desired value.

5 - Click on the button **Apply**. The criteria defined is then displayed in the list of the group **Request**.

6 - Repeat steps 2 to 5 for each characteristic on which you want to apply a selection criteria. The command button **Default** allows to come back to the default request, i. e. the request used to build the original list.

7 - Click on **OK**. The ***Filter*** dialog box disappears and the list of entities of the **Entity manager** window is updated according to the request that you have built.



Example -14 *Filter* dialog box

View the links between entities

The list of entities in the **Entity manager** window is known as a tree view control list as it offers a hierarchical view of the project's entities based on the links between them. Each level of this hierarchy is represented by a vertical alignment slightly shifted to the right in relation with the previous level.

The entities giving access to other levels are preceded by a box containing the symbol - or + depending whether the following levels are displayed or not. To display the levels, click on +. To hide the levels, click on -.

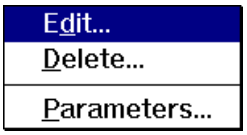
By selecting the appropriate option in the group **Tree view,** the links may be viewed in two different manners :

- Descending view : by reading the tree from the left to the right from one entity, you view the dependent entities, i. e. those « which were generated from the entity ». To get this view, select the option **Descending**.
- Ascending view : by reading the tree from the left to the right from one entity, you view the entities on Which the entity depends, i. e. those « from which it was generated ». To get this view, select the option **Ascending**

Modify the label and/or the description of an entity

You can modify the label and the description of a non-constant entity. In fact, the entities contained directly in the current project can be modified (entities coded in black).

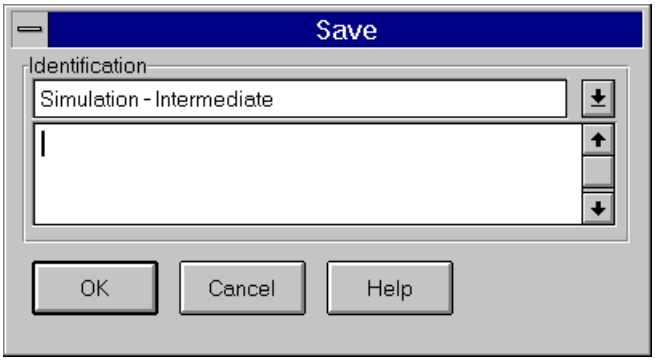
1 - In the list of entities of the **Entity manager** window, double-click on the entity for which you want to change the label and/or the description. If the operation is not allowed for this entity, a sound signal is emitted ; otherwise, the **Save** dialog box will appear.
Remark : The same result can be obtained by selecting the entity with the right mouse button and choosing the option *Edit...* in the pop-up menu displayed on the screen. If this option appears shaded, it means that the operation is not allowed for this entity.



2 - In the upper portion of the group **Identification**, change the name of the entity as needed.

3 - In the lower portion, change the description of the entity as desired.

4 - Click on **OK**.

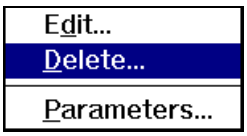


Example -15 Save dialog box

Delete an entity from the project

Under certain conditions, the non-constant entities can be deleted from the database of the project. In fact, the entities contained directly in the current project can be deleted (entities coded in black).

1 - In the list of entities of the **Entity manager** window, select the entity that you wish to delete and click with the right mouse button ; a pop-up menu will appear, comprising the item **Delete...** If this item is shaded, it means that the entity selected cannot be deleted.



2 - Select the item **Delete....** If other entities depend on the entity selected, the latter cannot be deleted before all the entities which depend on it have been deleted (see *Hints* below). In this case, a message will inform you of the situation. Otherwise, the **Delete** dialog box will ask you to confirm the delete command.

3 - Click on **Yes.** The entity is deleted from the database and the list is updated. If you click on **No**, the command is canceled.

Hints:

1 - To identify the dependent entities, display a descending list, as explained in the section *View the links between entities*.

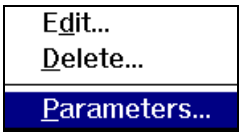
2 - To select several entities to be deleted, maintain the *Ctrl* key pressed down and click on each entity with the left mouse button. Then, follow steps 1 to 3. You erase them all in one operation by clicking on **All** in the delete dialog box. If you click on **Cancel,** the operation is canceled. You can also press down the **Shift** key to select a block of several adjacent entities.

3 - Once the entity(ies) to be deleted have been selected, you can simply press the delete key rather than using the pop-up menu.

Display information on the characteristics of the entity

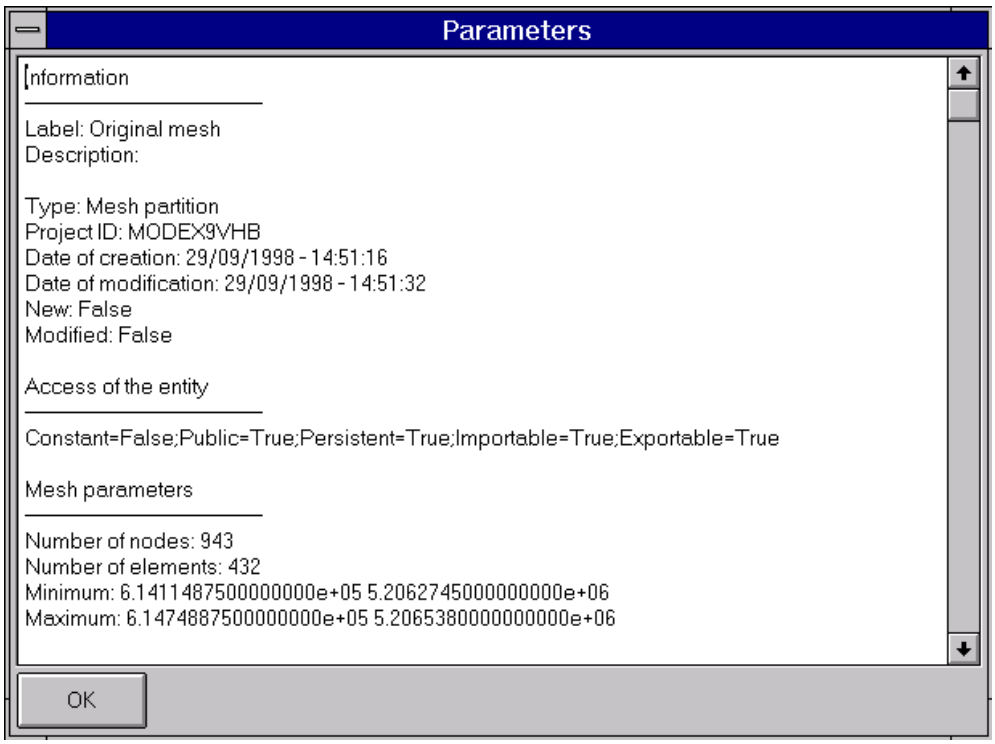
The upper right portion of the **Entity manager** window provides certain information concerning the entity selected in the list of project's entities. To obtain more detailed information on the characteristics of the entity, follow these steps :

1 - In the list of entities, click with the right mouse button on the entity for which you want to consult the characteristics. A pop-up menu will appear.



2 - In this menu, select **Parameters....** The **Parameters** window is displayed, providing a list of the characteristics of the entity selected. It is noteworthy that these characteristics are the same as those on which you can establish selection criteria (see *Display a list of the entities on selected* criteria).

3 - After consultation, click on **OK** to close the **Parameters** window.



Chapter 4 Controlling displays

In the software, modeling operations are conducted in a graphic environment allowing to view data under different forms in the display windows. When a graphic element is displayed in a window, a layer of information is added. To maximize the use of the information displayed, it is essential to be able to control the display formats.

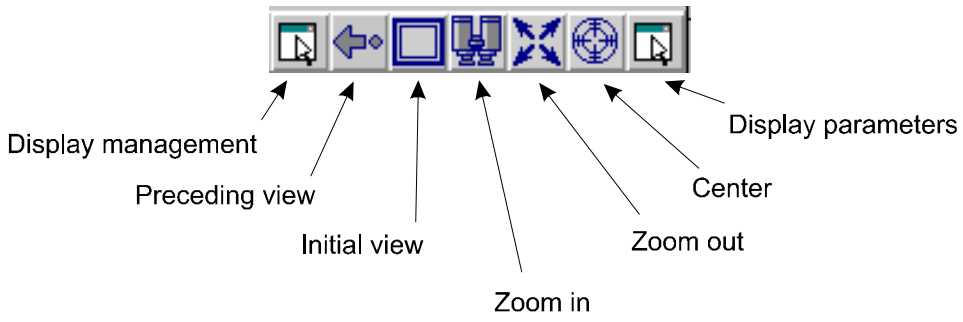
The software allows you to :

- Managing the graphic elements
- Controlling the view
- Printing the content of a window

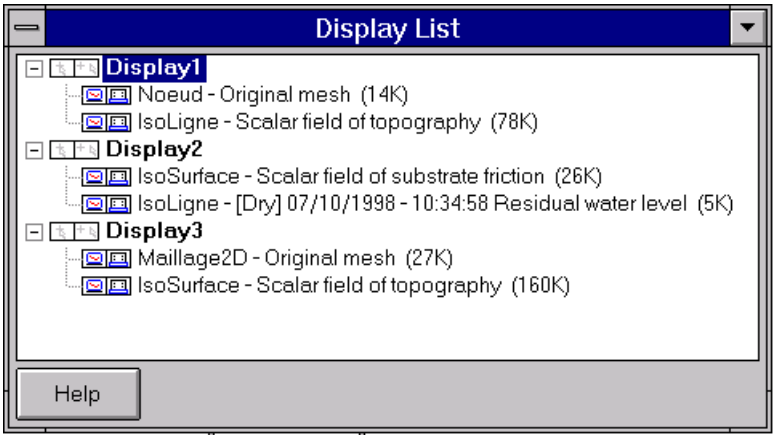
Managing the graphic elements

Several graphic elements can be displayed in a single display window ; therefore, it can be useful to manage each element individually.

When a display window is open, a tool bar controlling the display parameters appears.



In the display control tool bar, the button opens the management interface for the display list.


Display List
Display1
Noeud - Original mesh (14K)
IsoLigne - Scalar field of topography (78K)
Display2
IsoSurface - Scalar field of substrate friction (26K)
IsoLigne - [Dry] 07/10/1998 - 10:34:58 Residual water level (5K)
Display3
Maillage2D - Original mesh (27K)
IsoSurface - Scalar field of topography (160K)
Help

Example Chapter 4-1 Display list pop-up window

This interface offers a tree view of the opened windows with their respective graphic elements displayed within.

Management functions can be divided into three categories :

1 - Functions related to windows

2 - Functions related to graphic elements

3 - Functions related to groups of windows

The functions related to windows are :

- Opening a window
- Closing a window
- Activating a window
- Renaming a window
- Changing the cursor for a window

The functions related to the graphic elements inside a window are :

- Displaying a graphic element [screen]
- Displaying a graphic element [printer]
- Deleting a graphic element
- Changing the order of display of the elements
- Moving an element to another window
- Modifying the display parameters of an element
- Saving a graphic element
- Loading a graphic element previously saved
- Displaying the legend associated to a graphic element

The functions related to groups of windows are :

- Creating a group
- Closing a group
- Renaming a group
- Adding a window to a group
- Extracting a window from a group.

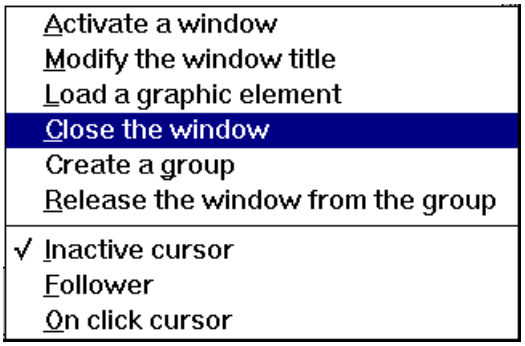
Opening a window

Before any display is possible, a window must necessarily be opened. To open a window, click on **Create** in the ***Windows*** menu.

A new display window will then appear overlying other windows.

Closing a window

To close a display window, click on the outline of the target window with the right button of your mouse. A pop-up menu will appear.



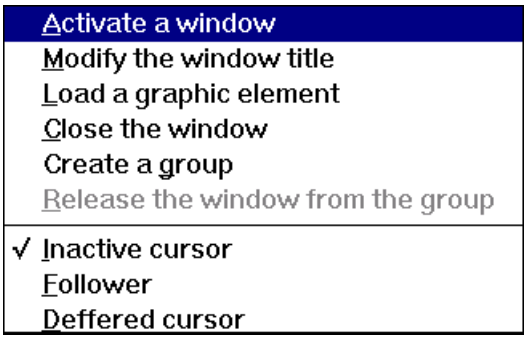
Select ***Close the window*** in this menu.

Activating a window

In order to display a new graphic element in a window, the window must first be activated.

Several different means are available to activate a window :

- Click on the title bar of the window and it will be automatically activated and will overlay other opened windows.
- Select the appropriate window in the **Window** menu where all opened windows are listed.
- Or finally, click with the right button on the outline of the target window in the display list management window. A pop-up menu will appear.

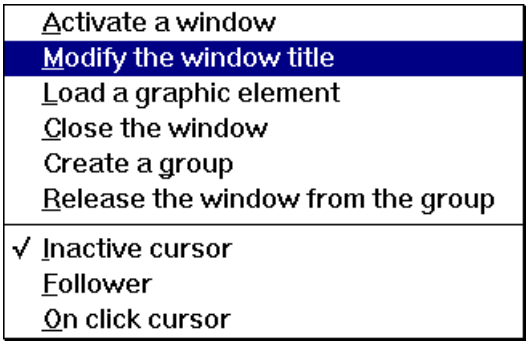


Select ***Activate the window*** in this menu.

Renaming a window

When several windows are opened, it can be useful to have a specific name for each. By default, the software names each window by a number : Display :# where # is the number.

To rename a window, click on the target window with the right button of your mouse. A pop-up menu will appear.



Select ***Modify the window title***. A dialog box will appear and you can type the chosen name.

Click **OK** to confirm or **Cancel** to cancel the operation.

Changing the cursor for a window

Three types of cursor are available for a window :

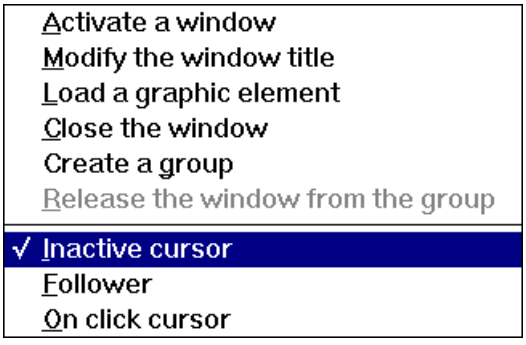
- the normal cursor
- the follower
- the on click cursor

The normal cursor is the regular mouse arrow overlying the content of the window.

The follower displays a special cursor over the selected window when the mouse pointer travels over ***another*** display window which has been activated. This allows to view a specific portion of the domain in one window, while knowing the exact location within the whole domain in the selected window.

The On click cursor is actually a variant of the preceding cursor which displays a special cursor at every mouse click in another window.

To select a cursor type, click on the target window with the right button of your mouse. A pop-up menu will appear.



Select the desired cursor in the lower portion of this menu : Normal cursor, follower or On click cursor. The current selection will be checked off.

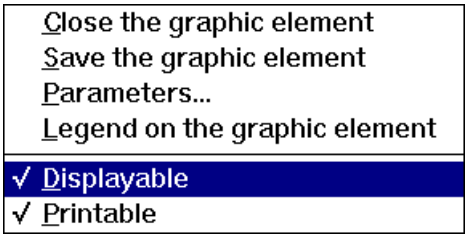
Displaying a graphic element [screen]

When several graphic elements are overlaid in a display window, details of some elements may be concealed by others. Temporarily hiding some elements will allow to view the details of interest.

To hide/display a graphic element, follow these steps :

1 - In the display list, click with the right button of your mouse on the element to be hidden or displayed. A pop-up menu will appear.

2 - In this menu, activate or deactivate the *Displayable* option.



Hint :

- A quick way to reach the same result is to double-click on the element to display/hide in the list.

Notes :

- You can view the status of a graphic element [displayed/hidden] directly on the line for this element in the display list management window. The first icon in front of the name represents a screen. If it is shaded, the corresponding graphic element is hidden. Actually, the entire line becomes shaded when the graphic element is not displayed on screen.

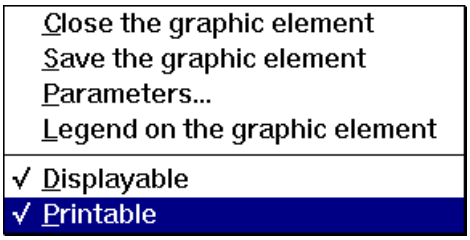
Displaying a graphic element [printer]

The software offers the user some additional flexibility for printing graphic elements. It is possible to print only selected elements without necessarily removing the non-selected elements from the display screen. Also, a graphic element does not have to be necessarily displayed on screen to be printed.

To display/hide a graphic element for printing, follow these steps :

1 - In the display list, click with the right button of your mouse on the element to be hidden or displayed. A pop-up menu will appear.

2 - In this menu, activate or deactivate the *Printable* option. The element appears or disappears for printing.



Notes :

- You can view the status of a graphic element [displayed/hidden] directly on the line for this element in the display list management window. The second icon in front of the name represents a printer. If it is shaded, the corresponding graphic element is not for printing.

Deleting a graphic element

You can remove a graphic element from a window by simply deleting it from the display list.

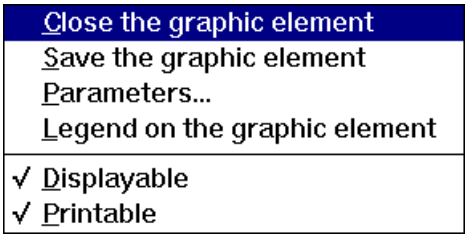
However, this function applies only to graphic elements known as **static**, i.e. elements which originated from the static display. This category of graphic elements includes meshes, isolines, isosurfaces, vectors, nodes and profiles.

The **non static** graphic elements cannot be removed from the display by this simple technique. This category includes the whole range of partitions and the cartographic dressing options. These **interactive** graphic elements must be handled within their respective modes.

To delete a graphic element, follow these steps :

1 - In the display list, click with the right button of your mouse on the element to be deleted. A pop-up menu will appear.

2 - In this menu, select **Close the graphic element**. The element is removed from the window and the display list is updated.



Hint :

- A graphic element can be removed from a window by selecting it and pressing **Delete** on the keyboard.

Notes :

- The entity representing this element is not removed from the database and can be displayed again, when needed.

Changing the order of display of the elements

Initially, the graphic elements in a display window are overlaid in order of appearance, the last element selected forming the top layer. The display list reflects this situation. The order of the elements in the list corresponds to their order of display in the window.

The order of display greatly affects the visibility of an element, since the overlying elements may conceal the details of the element.

You can change at will the order of display of the elements by following these steps:

1 - In the display list, click with the left button of your mouse on the graphic element to move.

2 - Hold the left button down, move the cursor, which will then change form, to point at the position in the list where you wish to move the selected element (drag and drop technique).

3 - Release the button. The elements are now displayed according to the new order and the display list is updated. The cursor returns to the initial form.

Moving an element to another window

Graphic elements can be moved from window to window. It is also possible to copy and paste a graphic element to another window.

To move a graphic element, follow these steps :

1 - Use the drag and drop technique by maintaining the left button down after selecting the graphic element to be moved.

2 - Drag the element to the **destination** window. The cursor changes form during this operation. Upon reaching the **destination** window, release the button.

3 - The graphic element is then moved from the **source** window to the **destination** window. Both windows are immediately displayed again.

Restrictions :

- Only the **static** graphic elements can be moved from window to window. A special cursor will be displayed if the transfer is not allowed.

Notes :

- To **copy** a graphic element from window to window, i.e. leave the graphic element inside the **source** window while pasting this element into the **destination** window, simply follow steps 1-3 while pressing down **Control** (Ctrl) on your keyboard.

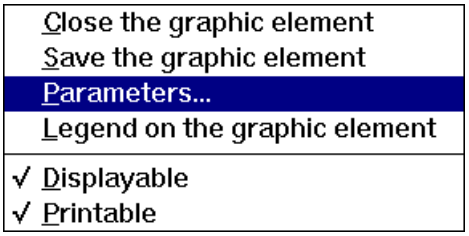
Modifying the display parameters of an element

In order to display a graphic element in a window, you have to be in the static display mode. A series of buttons allow the display of static graphic elements. The required information is selected in a dialog box and the display is then possible.

To modify the display parameters for a given graphic element, the dialog box can be reopened and the parameters modified. However there is one restriction ; you cannot change the data on which the display was specified.

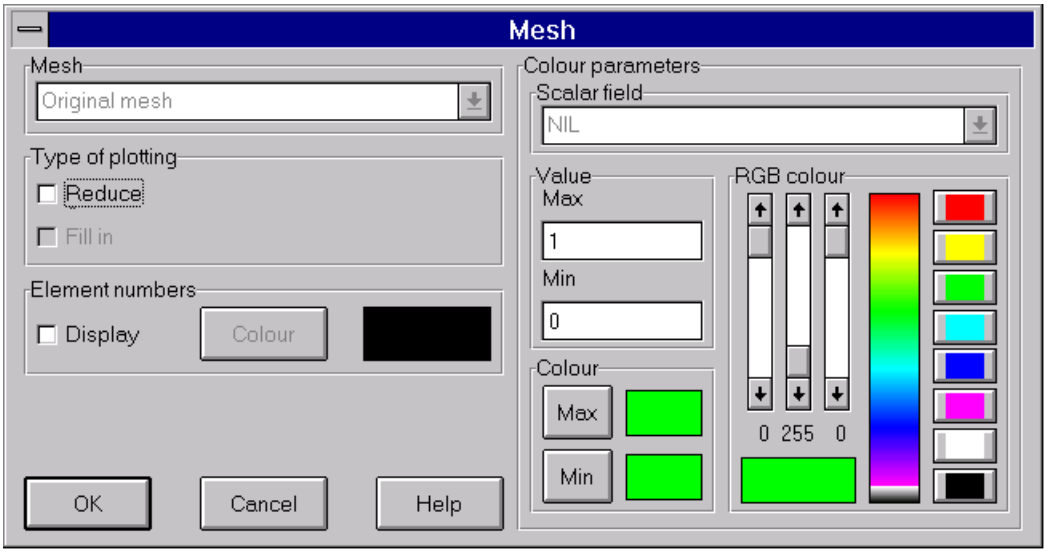
If, for example, you wish to modify the display parameters for a mesh, follow these steps :

1 - In the display list, click with the right button of your mouse on the graphic element representing the mesh. A pop-up menu will appear.



2 - Click on **Parameters...** and a dialog box will appear.

3 - Change the parameters according to your needs and click on **OK** to confirm the changes or click on **Cancel** to delete the changes. The modifications are immediately applied and the window is displayed again.



Restrictions :

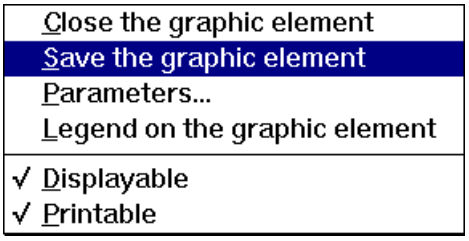
- Currently, modifications of display parameters are only possible for meshes, isolines, isosurfaces, vectors and nodes.
- For the other types of graphic elements, they must be deleted from the display and the operation must resume from the beginning.

Saving a graphic element

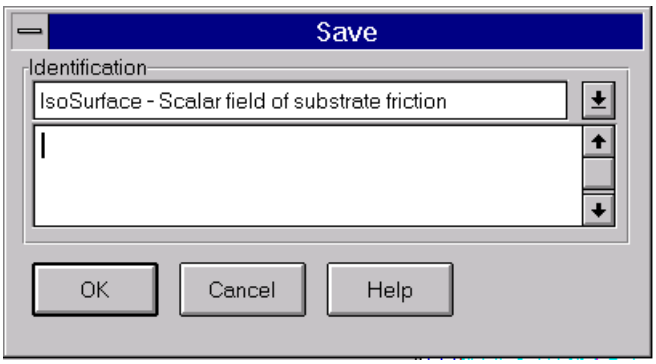
In the database, the static graphic elements are GR (Graphic Representation) type entities. By default, these entities are non-persistent, i.e. they are not saved on disk and exist only in the system's memory. Using the display list, you can save these entities in order to use them in further work sessions on the project.

To save a graphic element, follow these steps :

1 - In the display list, click with the right button of your mouse on the graphic element to be saved. A pop-up menu will appear.



2 - In this menu, click on **Save the graphic element**. The **Save** dialog box is displayed.



3 - Type the label you wish to affix to this graphic element in the upper part of the dialog box.

4 - If desired, type a description in the lower part of the dialog box.

5 - Click on **OK** to confirm the operation or on **Cancel** to cancel the operation.

Notes :

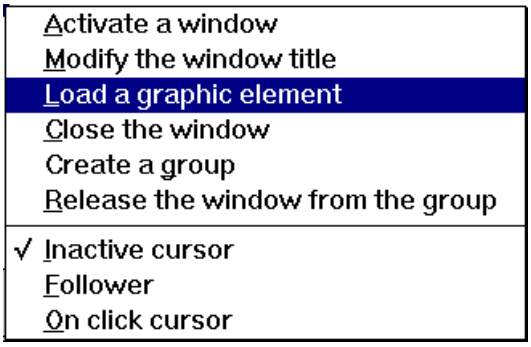
- To redisplay the element you have just saved, you must use the function Load a graphic element. (See *Loading a graphic element previously saved*.)

Loading a graphic element previously saved

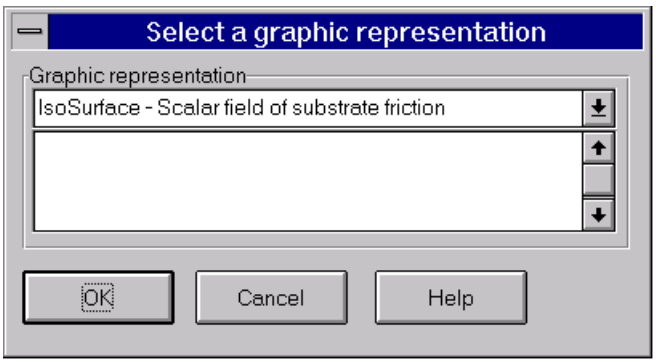
Certain **static** graphic elements can be loaded at will for display in a window. The graphic elements which can be loaded are those which have been previously saved in the database. (See *Saving a graphic element*.)

To load a graphic element, follow these steps :

1 - In the display list, click with the right button of your mouse on the outline representing the ***selected window***. A pop-up menu will appear.



2 - In this menu, click on **Load a graphic element**. The **Graphic Element Selection** dialog box will appear.



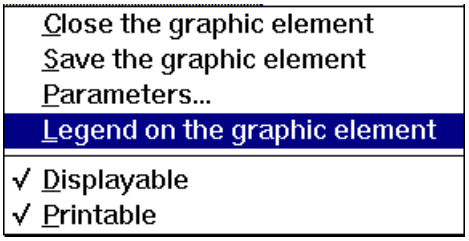
3 - In the drop-down list in the dialog box, select the graphic element you wish to load. Its description, if there is one, will then appear in the lower part of the box.

4 - Click on **OK**. The graphic element is displayed in the window and a line of descriptive text is added at the end of the display list. If you do not want to load the graphic element, click on **Cancel**.

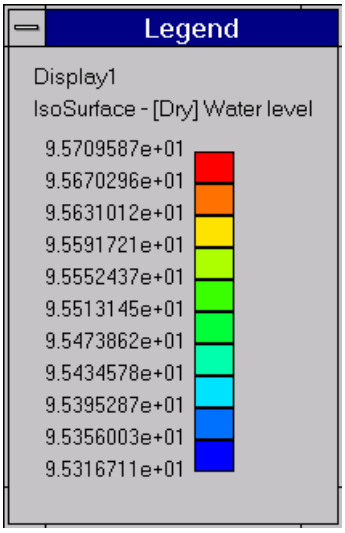
Displaying the legend associated to a graphic element

When creating a static graphic element, you can index the display colour on a selected scalar field. (See the literature on static display.) A relation is then created between the different display colours and values (or value intervals). This relation can be viewed as a legend that you can display by following these steps :

1 - In the display list, click with the right button of your mouse on the element for which you want to display the legend. A pop-up menu will appear.



2 - In this menu, click on **Legend of a graphic element**. The legend will appear on the screen.



Notes :

- It is only possible to display a legend for the graphic elements representing index-linked values such as isolines and isosurfaces.

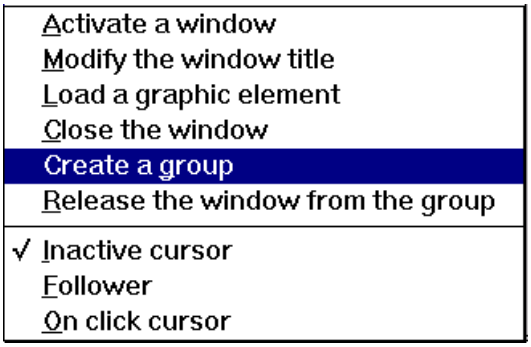
Creating a group

The software offers the possibility of linking various display windows together, allowing for a single movement reaction to display modifications.

For example, if two windows are linked together and a zoom in is ordered on one of them, the same zoom in will also affect the second window. This applies to all view control commands. (See *Controlling the view*).

To create a group, follow these steps :

1 - In the display list, click with the right button of your mouse on a window outline. A pop-up menu will appear.



2 - In this menu, click on **Create a group**. A group is then created and the selected window is added to the group.

Notes :

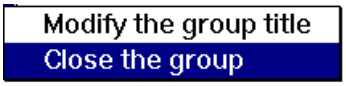
- If the selected window is already part of a group, it will be deleted from that group to be added to the new group.
- To add windows to the new group, see *Adding a window to a group*.

Closing a group

After creating a group, it is still possible to turn back by closing it. To create a group, see *Creating a group*.

To close a group, follow these steps :

1 - Click on the target group with the right button of your mouse. An pop-up menu will appear.



2 - In this menu, click on **Close the group**. The group is then deleted and the windows which were part of it return to the off-group window space.

Hint :

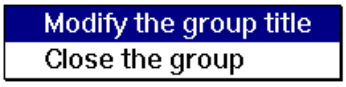
- To quickly close a group, you can select the group and press **Delete** on the keyboard.

Renaming a group

For practical purposes, you can give specific names to groups you are creating.

To rename a group, follow these steps :

1 - Click on the target group with the right button of your mouse. A pop-up menu will appear.



2 - In this menu, click on **Rename the group**. A dialog box will appear.

3 - Type in the new name and click on **OK** to confirm the change or on **Cancel** to keep the original name of the group.

Adding a window to a group

After creating a group, it is possible to add or extract at will windows from this group.

To add a window to a group, follow these steps :

1 - Click on the target window with the left button of your mouse.

2 - Drag the selection to its destination group. The cursor changes aspect during the drag operation.

3 - When the destination group is reached, release the mouse button (drop operation). The window is added to this group.

See also :

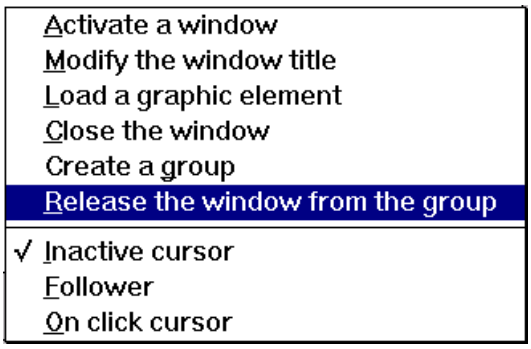
- See *Creating a group*
- See *Extracting a window from a group*

Extracting a window from a group

After creating a group, it is possible to add or extract at will windows from this group.

To extract a window from a group, follow these steps :

1 - Click on the target window with the left button of your mouse. A pop-up menu will appear.



2 - In this menu, click on **Release the window from the group**. The window returns to the off-group window space.

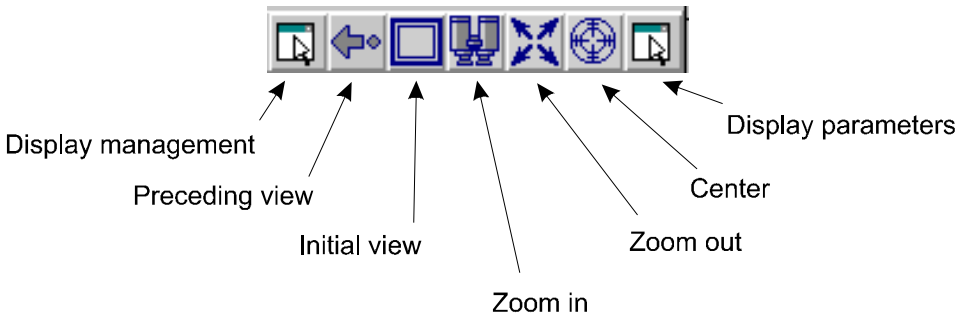
Controlling the view

In many ways, a display window may be considered as a rectangular opening through which you can « see » your project, or, more precisely, the graphic representations of some of the data. By « moving » the opening, you change the view it provides ; for example, getting it closer is like zooming in, pivoting it is like making a rotation...

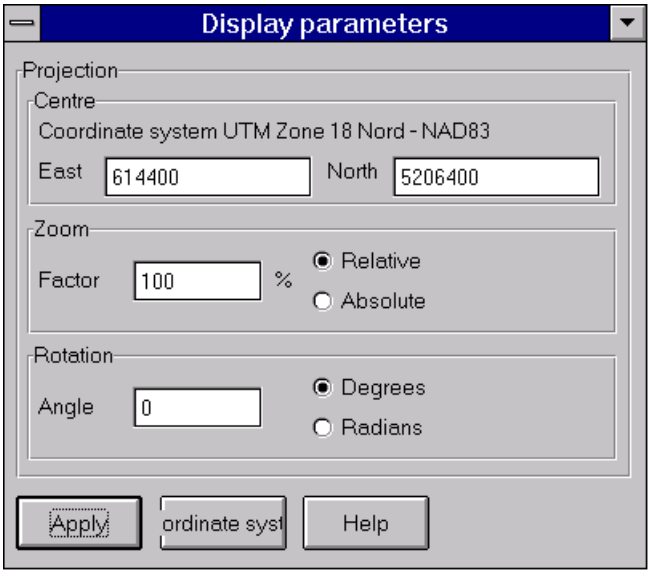
The software provides functions to control the view offered by a display window. Thus you can :

- Zooming in
- Zooming out
- Re-centering the view
- Making a rotation
- Returning to the preceding view
- Returning to the initial

A function tool bar becomes accessible as soon as a display window is open.



These functions can be used individually or in combination with each other through the **Display parameters** window accessible by clicking on the function button located in the lower right corner of the screen.



Example Chapter 4-2 Display parameters pop-up window

Zooming in

To view with more details a specific area of your project, the software provides the possibility to enlarge it (zoom in). This enlargement can either be **relative**, i.e. originating from the current view, or **absolute**, i.e. calculated from the initial view.

To make a relative zoom in, follow these steps:

1 - Activate the target display window by clicking on it or by selecting it in the **Window** menu.

2 - Click on the function button located at the bottom of the screen. A cross indicating the center of the active window appears as a reference point.

3 - Place the cursor at the center of the area you wish to enlarge. The cursor takes the form of a cross to facilitate localization.

4 - With the left button of the mouse pressed down, drag the cursor. A rectangle defining the area to enlarge extends or shrinks with the moving cursor.

5 - Once the zone to enlarge has been properly defined with the rectangle, release the mouse button.

Notes :

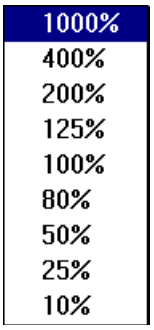
- Prior to step 4, it remains possible to cancel the operation by clicking again on the function button .

- This method is actually a combination of a zoom in and of a re-centering operation (see *Re-centering the view*).

To make an absolute zoom in, follow these steps:

1 - Activate the target display window by clicking on it or by selecting it in the **Window** menu.

2 - Click on either one of the function buttons or . A pop-up menu showing predefined zoom factors will appear on the screen.



3 - Select the zoom percentage wanted.

Another method to zoom in is to use the **Display parameters** (Example Chapter 4-2) :

1 - Activate the target display window by clicking on it or by selecting it in the **Window** menu.

2 - If the **Display parameters** pop-up window is not already displayed, click on the function button (lower right corner of the screen) to make it appear.

3 - In the **Zoom** group, click either on **Relative** or **Absolute** depending upon the type of zoom wanted.

4 - In the **Factor** field, type in the wanted zoom factor in percentage.

5 - Click on **Apply**.

Zooming out

For a reduced view of the project, the software provides a zoom out function. As with the zoom in (see Zooming in) the reduction can either be **relative**, i.e. originating from the current view, or **absolute**, i.e. calculated from the initial view.

To make a relative zoom out, follow these steps:

1 - Activate the target display window by clicking on it or by selecting it in the **Window** menu.

2 - If the **Display parameters** pop-up window is not already displayed, click on the function button (lower right corner of the screen) to make it appear.

3 - In the **Zoom** group, click either on **Relative** or **Absolute** depending upon the type of zoom wanted.

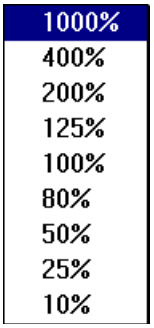
4 - In the **Factor** field, type in the wanted zoom factor in percentage.

5 - Click on **Apply**.

To make an absolute zoom out, follow these steps:

1 - Activate the target display window by clicking on it or by selecting it in the **Window** menu.

2 - Click on either one of the function buttons or . A pop-up menu showing predefined zoom factors will appear on the screen.



3 - Select the zoom percentage wanted.

Re-centering the view

For an interactive modification of the center of the view, follow these steps:

1 - Activate the target display window by clicking on it or by selecting it in the **Window** menu.

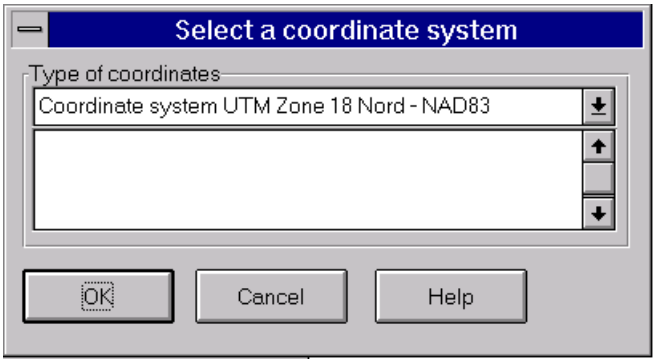
2 - Click on the function button located at the bottom of the screen. A cross indicating the center of the active window appears as a reference point.

3 - Place the cursor, now in the form of a cross, at the position selected as new center and click. The reference point disappears and the display is updated with a new center for the window.

To change the center of the view using the **Display parameters** window, follow these steps:

1 - If the **Display parameters** pop-up window is not already displayed, click on the function button (lower right corner of the screen) to make it appear.

2 - If the coordinate system in use is not the one you wish to use, click on **Coordinate system**. The **Coordinate system selection** dialog box appears.



3 - In the drop-down list, choose the type of coordinate system you wish to use to indicate the center and click on **OK.** The dialog box disappears and the type of coordinate system selected is displayed in the section **Center** of the parameters window.

4 - In the section **Center**, enter the **X** and **Y** coordinates of the center, using the syntax of the chosen coordinate system.

5 - Click on **Apply**.

Making a rotation

It is possible to rotate the study domain. By default, the domain is always orientated with the north at the top.

However, if a river flows from southwest to northeast, it may be convenient to rotate the domain by 45 degrees so that the river flows from left to right.

To rotate the domain, follow these steps :

1 - Activate the target display window by clicking on it or by selecting it in the **Window** menu.

2 - If the **Display parameters** pop-up window is not already displayed, click on the function button (lower right corner of the screen) to make it appear.

3 - In the group **Rotation,** click on either **Degrees** or **Radians** depending on the angle measure units you wish to use.

4 - In the field **Angle**, enter the rotation angle in the units previously selected.

5 - Click on **Apply**.

Returning to the preceding view

At any given time, you can return to the preceding view by clicking on the function button located at the bottom of the screen. However, this operation works only backward, i.e. the current view is deleted to be replaced by the preceding view.

Returning to the initial view

At any given time, you can return to the initial view of your project, i.e. the view specified in the project parameters.

To return to the initial view, click on the function button located at the bottom of the screen. The window then displays the initial view.

Printing the content of a window

Whatever is displayed in a window can be printed on paper using the available printers, or sent to a Postscript (PS), an Encapsulated Postscript (EPS), or a bitmap (BMP) file. The software provides an interface allowing to specify the printing destination (printer or file), the size and orientation of the image, as well as a number of setup parameters such as a frame and a title.

You can control what you wish to print from a window ; see *Displaying a graphic element [printer].*

To print the content of a window, follow these steps :

1 - Activate the display window from which you want to print by clicking on it, or by selecting it in the **Window** menu.

2 - Click on **Print...** in the **Project** menu. The **Printing** dialog box appears.

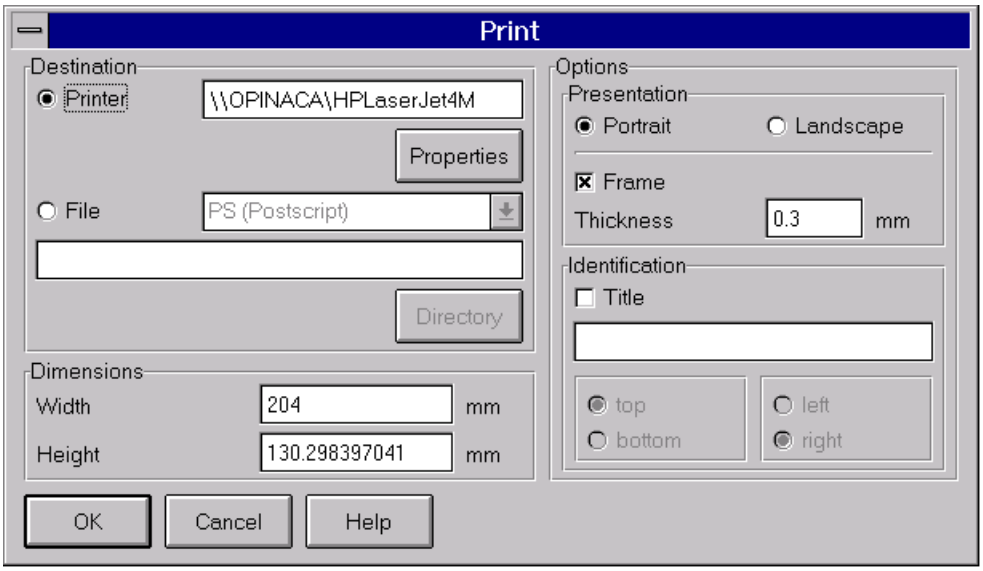
3 - In the section **Destination**, select the printer or file where printing will occur. (See *Specify printing destination*)

4 - In the section **Dimensions,** specify the image size. (See *Specify image printing size*)

5 - In the section **Setup**, specify the orientation of the image and, if wanted, its framing. (See *Specify printing setup*)

6 - If you wish, you can give a title and a position for it in the section **Identification.**

7 - Click on **OK**.



Example Chapter 4-3 Printing dialog box

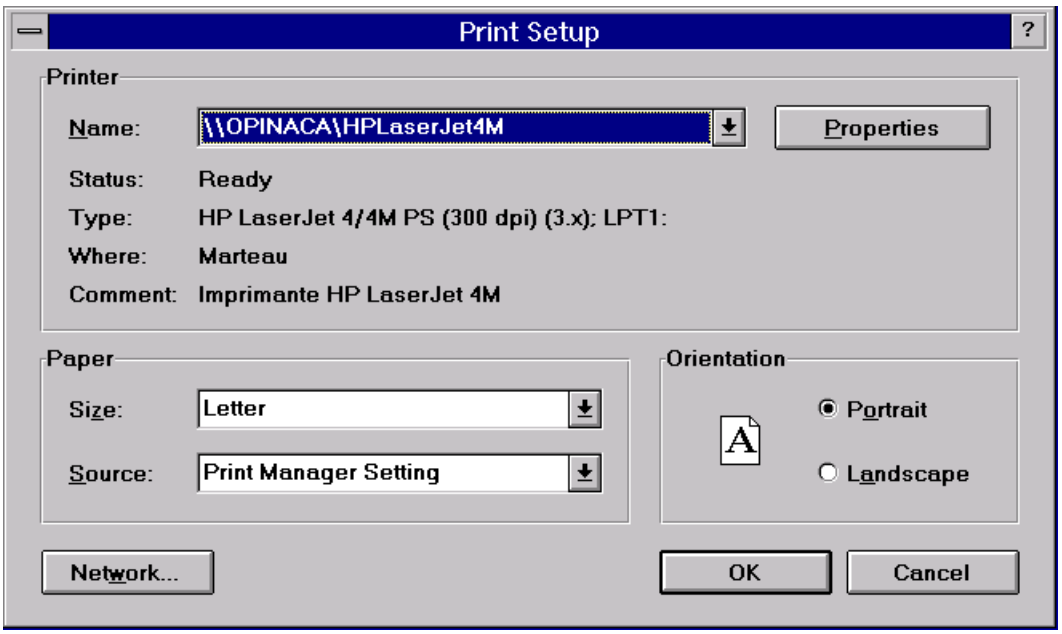
Specify printing destination

The section **Destination** in the **Printing** dialog box informs the software on which printer or file the job should be sent to.

To select a printer, follow these steps :

1 - Click on **Printer**. The name of the default printer is displayed in the field to the right of this option.

2 - You can select a different printer and change the parameters by clicking on Properties. This button leads to the **Print setup** dialog box of the operating system.



To select a file, follow these steps :

1 - Click on **File**.

2 - In the drop-down list to the right of this option, select the type of destination file.

3 - In the edit field, enter the file name and the access pathway. You can also explore the directories by clicking on **Directory :**

a) In the section **Directory,** double-click on the directory where the print file should be sent**.** The files (of the type selected in step 2) contained in this directory are displayed in the section **Files.** As you are making selections, the current access pathway is displayed in the section **File**.

b) In the section **File**, enter the file name. You can also select a file already available in the section **Files.** In this case, the software will ask you to confirm that you want the file replaced.

c) Click on **OK**.

Specify image printing size

The size of the printed image is automatically determined by its orientation on the page (see *Specify printing setup*) and its proportions in the display window. The size can be modified in the section **Dimensions** of the **Printing** dialog box.

Specify printing setup

The group **Options** in the **Printing** dialog box allows you to modify setup parameters such as orientation, frame, and title :

1 - In the section **Setup**, click on either **Portrait** or **Landscape** to select the orientation of the image on the page. These options are also available in the **Print setup** dialog box accessible by clicking on Properties in the section **Destination** (see *Specify printing destination*).

2 - To frame the image, click on **Frame** and type the frame thickness in points in the field **Thickness**.

3 - If desired, select the **Title** check box in the **Identification** section and type a title in the edit field. The position of the title on the sheet is determined by selecting the options **upper** or **lower**, and **left** or **right**.

Chapter 5 Data display

The software provides a series of display functions to facilitate validation and interpretation of the data in your project.

You can display your data in the display windows and add text, legends, scale bars, etc. to help interpretation.

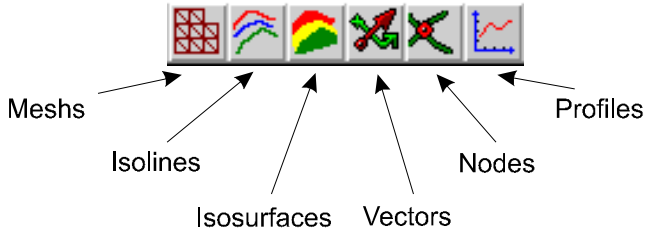
The software allows you to :

- Data display
- Cartographic dressing

Data display

Data can be displayed in various formats depending upon the type of data. The graphic elements created this way are known as **static** because they cannot be modified ; therefore the software unit providing these functions is named the static display. To get access to this unit, click on *Static* in the **Display** menu. The following tool bar will then appear at the bottom of the screen :

Figure 1 Static display tool bar



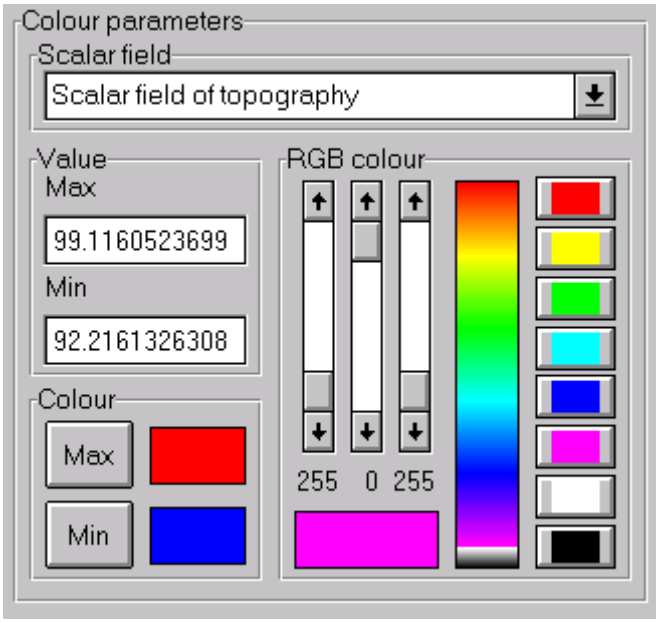
Each button opens a dialog box to specify display parameters. Thus you can :

- Display a mesh
- Display isolines
- Display isosurfaces
- Display vectors
- Display the mesh nodes
- Display profiles

These dialog boxes have a common section **Colour parameters** used to control the display colours. This section is described separately in *Colour parameters setup.*

Colour parameters setup

The static display dialog boxes share a common part used to control the display colours :



Example -4 Colour parameters page

Not only do colours have an aesthetic function, but they also provide broader information within a single display by indexing the colour of the layout on a scalar field. For example, you can trace a vector the length of which is proportional to the speed unit and the colour is a function of topography. The only colour parameters dialog box which differs slightly from the others is that of isosurfaces. Colours are already an integral part of the layout and cannot be indexed on a different scalar field than that used to define isosurfaces. However, the colour selection and allocation for isosurfaces are similar to other dialog boxes. The procedure is as follows :

1 - Click on a field in the drop-down list of the section **Scalar field**. The **Max** and **Min** edit fields in the section **Value** are given the maximum and minimum values of the chosen scalar field.

Note : The list contains the scalar fields associated with the mesh previously selected in the left section of the respective dialog boxes. It is not essential to index the colours when displaying meshes, vectors, or nodes ; simply click the **NIL** value in the list.

2 - If you wish, you can modify the range of values by changing the values of the **Max** and **Min** edit fields in the section **Value.**

3 - Determine the colour range for the layout by giving a colour to the maximum value and a colour to the minimum value. Select the desired colour in the section **Colour RGB** using one of the following methods :

- by varying the input of red, green and blue using the three trackbars.
- by clicking on the desired colour in the colour spectrum bar. The *RGB* trackbars are then automatically adjusted to the colour selected.
- by clicking on one of the predefined colours. Again *RGB* trackbars are then automatically adjusted to the colour selected.

In each case, the selected colour will appear in the box located below the trackbars. Then simply click on the appropriate button (**Max** or **Min**) to apply your selection. The colour then appears in the box related to the activated button.

The layout colours will vary evenly in the range [minimum colour, maximum colour]. As the **Max** and **Min** edit fields can be modified, the following rules apply when the max and min values of the scalar field differ from the max and min values that you have entered :

- if the selected maximum or minimum values lie beyond the extreme values of the scalar field, only a part of the colour range will be used to represent the data.
- if the selected maximum or minimum values lie within the extreme values of the scalar field, the entire colour scale will be used to represent the data within the specified range. Data beyond these limits will be given the colour of either the maximum or the minimum value, accordingly.

Display a mesh

Displaying a mesh allows you to view the distribution of its elements. You can also improve the graphic representation by indexing the display colour in one of the scalar fields related to it.

Prerequisite : The following conditions must be fulfilled before a mesh can be displayed :

- a project must be loaded (see the documentation on project management.)
- at least one display window must already exist (see the documentation on display control.)

To specify the display parameters for a mesh, follow these steps:

1 - After clicking on the option **Static [...]** in the **Display** menu, click on in the function toolbar at the bottom of the screen. The **Mesh** dialog box will appear.

2 - In the drop-down list of the section **Mesh**, select the mesh to trace. This list includes all the meshes of the current project.

Note : Because of values given by default, the selected mesh can be displayed now by simply clicking on **OK**.

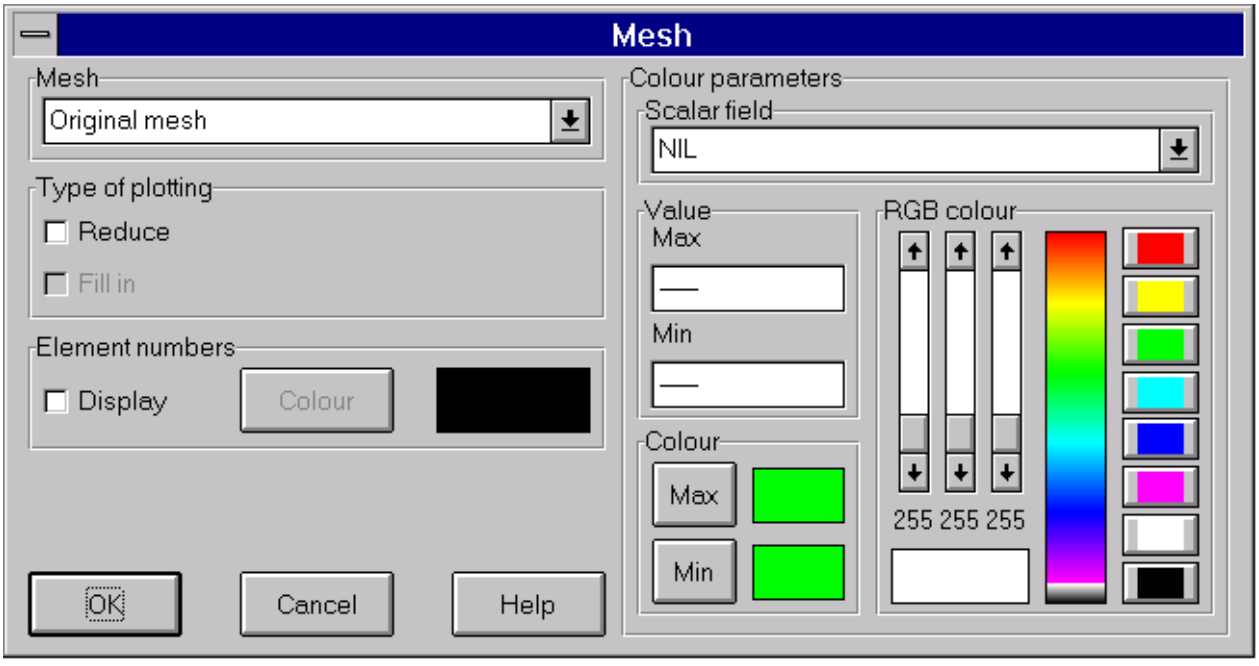
3 - If wanted, click on the radio button **Reduced** in the section **Type of layout** to trace the elements in a reduced format. (See *Display reduced elements*)

4 - If wanted, you can modify the parameters controlling the display of the numbers of the mesh elements in the section **Element numbers**. (See *Display element numbers*)

5 - If needed, you can modify the various parameters controlling the colours of the layout and their indexing in the section **Colour parameters**. (See *Colour parameters setup*)

6 - Click on **OK**.

Note: If, at step 5, you have decided to index the colours on a nodal value, you can display the legend of the colours used (See the section on the display of a legend in the documentation on Display control.)



Example -5 Mesh dialog box

Display element numbers

Internally, each element of the mesh is identified by a number. You can display the numbers by :

1 - Checking **Display** in the section **Element numbers**. The **Colour** button is then activated.

2 - Selecting the colour for the element numbers in the section **Colour RGB** (see *Colour parameters setup*).

3 - Clicking on **Colour.** The selected colour will appear in the box associated with the button.

Display reduced elements

By default, the type of layout for the mesh elements is non-reduced, i.e. the elements are connected by common edges. This type of layout illustrates the theoretical representation of the elements of a mesh, but it hides the blocks that could have been formed while generating the mesh.

The software provides the possibility of a reduced layout, in which the elements appear disconnected, which allows to detect any block present. Simply activate the radio button **Reduced** in the section **Type of layout** of the **Mesh** dialog box.

Figure -1 Reduced display compared with the normal display of elements

Normal display Reduced display

Display isolines

With this function, you can view the isolines of a scalar field, i.e. the lines linking up all the points of same value. An isoline layout provides broad information :

- the form and position of isolines illustrate the distribution of values in space.
- the distance between successive isolines gives the variation rate of values in space.
- the colour of the lines gives information on the value represented by any given isoline. The colours of the lines can also be indexed in another scalar field.

Prerequisite : Prior to the display of isolines, verify the following points :

- a project must be loaded (see the documentation on project management.)
- at least one display window must already exist (see the documentation on display control.)

To specify the isoline display parameters, follow these steps :

1 - After clicking on the option **Static [...]** in the **Display** menu, click on in the function toolbar at the bottom of the screen. The **Isoline** dialog box will appear.

2 - In the drop-down list of the section **Mesh**, choose the mesh used to generate the scalar field from which you want to display the isolines. This list includes all the meshes of the current project.

3 - In the drop-down list of the section **Scalar field**, select the field from which the isolines will be traced. This list contains scalar fields related to the mesh previously selected. The **Min** and **Max** edit fields are given the maximum and minimum values of the chosen scalar field.

Note : Because of values given by default, the isolines can be displayed now by simply clicking on **OK**.

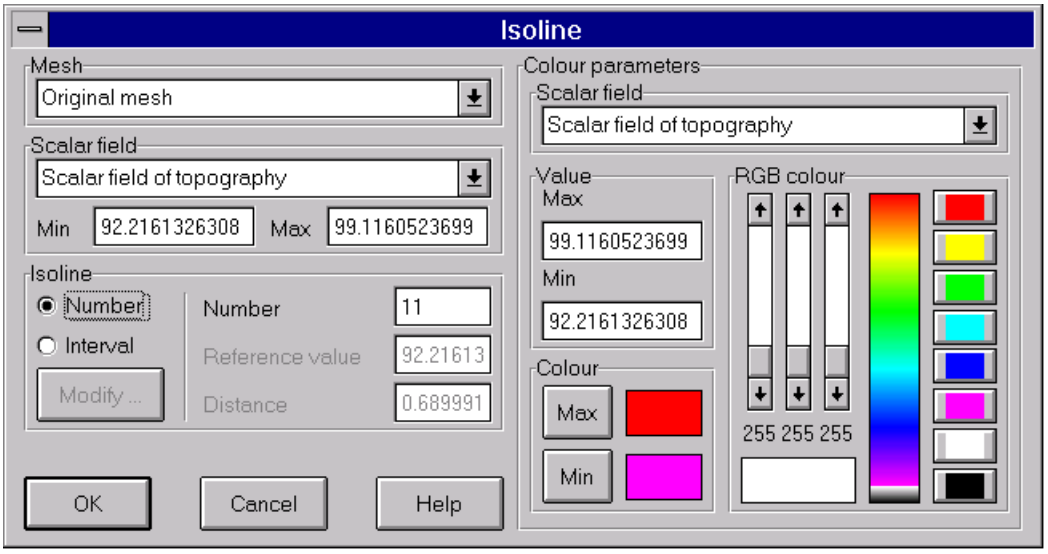
4 - In the **Min** and **Max** edit fields, you can change the limit values of the value range for which isolines will be traced.

5 - If wanted, you can modify the parameters controlling the number of isolines to trace in the section **Isoline**. (See *Specify the number of* isolines)

6 - If needed, you can modify the various parameters controlling the colours of the layout and their indexing on another nodal value in the section **Colour parameters**. (See *Colour parameters setup*)

7 - Click on **OK**.

Note: If, at step 6, you have selected different colours for the min and max values, you can display the legend of the colours used. (See the section on the display of a legend in the documentation on Display control.)



Example -6 Isolines dialog box

Specify the number of isolines

Drawing isolines can be done according to a fixed number of isolines or to a range. You can specify the parameters for either one of these two methods in the section **Isoline** of the **Isoline** dialog box:

Method 1 :

1 - Click on **Number**.

2 - In the edit field **Number,** enter the number of values to which isolines will be associated between the **Min** and **Max** values specified in the section **Scalar field**. By default, the number is set at 11.

The range between the **Min** (included) and the **Max** (included) will then be divided according to the following rules :

- If the specified **Min** or **Max** values lie within the range of the extreme values of the scalar field, the selected number of isolines will be traced, but only for the data included in the specified range.
- If the specified **Min** or **Max** values lie beyond the extreme values of the scalar field, the isolines will cover the whole set of data, but will be less than the number specified.

Method 2 :

1 - Click on **Range**.

2 - In the **Gap** edit field, enter the values corresponding to two successive isolines. The default value for the gap is calculated in such a way that, with the parameters by default, the result will be the same no matter which method is used.

3 - You can specify the value from which successive intervals will be calculated by adding the selected gap in the **Reference value** edit field. The reference value by default is the lowest value in the data set.

In this method, the following rules apply :

- If the specified **Min** or **Max** values lie within the range of the extreme values of the scalar field, the isolines will be traced for the data included between these limit values.
- If the specified **Min** or **Max** values lie beyond the extreme values of the scalar field, the isolines will cover the whole set of data.

The two methods being exclusive, the fields irrelevant to the chosen method are deactivated when selecting an option.

Display isosurfaces

With this function, you can view the isosurfaces representing a scalar field. An isosurface is a surface representing all the values within a given interval. An isosurface provides a variety of information:

- the form and position of isosurfaces illustrate the distribution of values in space.
- the width of isosurfaces gives the variation rate of values in space.
- the colour of each surface gives information on the values included in a given isosurface.

Prerequisite : Prior to the display of isosurfaces, verify the following points :

- a project must be loaded (see the documentation on project management.)
- at least one display window must already exist (see the documentation on display control.)

To specify the isosurface display parameters, follow these steps:

1 - After clicking on the option **Static [...]** in the **Display** menu, click on in the function toolbar at the bottom of the screen. The **Isosurface** dialog box will appear.

2 - In the drop-down list of the section **Mesh**, choose the mesh used to generate the scalar field from which you want to display the isosurfaces. This list includes all the meshes of the current project.

3 - In the drop-down list of the section **Scalar field**, select the field from which the isosurfaces will be traced. This list contains scalar fields related to the mesh previously selected. The **Min** and **Max** edit fields are given the maximum and minimum values of the chosen scalar field.

Note : Because of values given by default, the isosurfaces can be displayed now by simply clicking on **OK**.

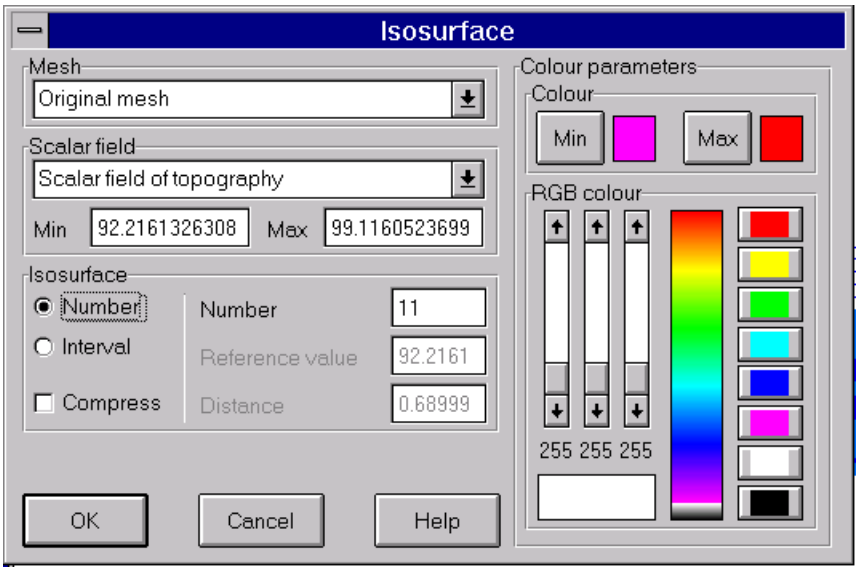
4 - In the **Min** and **Max** edit fields, you can change the limit values of the value range for which isosurfaces will be traced.

5 - If wanted, you can modify the parameters controlling the number of isolines to trace in the section **Isosurface**. (See *Specify the number of* isolines)Specify the number of isosurfaces

6 - If needed, you can modify the various parameters controlling the colours of the layout and their indexing on another nodal value in the section **Colour parameters**. (See *Colour parameters setup)Colour* parameters setup

7 - Click on **OK**.

Note: If, at step 6, you have selected different colours for the min and max values, you can display the legend of the colours used. (See the section on the display of a legend in the documentation on Display control.)



Example -7 Isosurfaces dialog box

Specify the number of isosurfaces

Drawing isosurfaces can be done according to a fixed number of isosurfaces or to a range. You can specify the parameters for

either one of these two methods in the section **Isosurface** of the **Isosurface** dialog box:

Method 1 :

1 - Click on **Number**.

2 - In the edit field **Number,** enter the number of values marking the boundaries of isosurfaces between the **Min** and **Max** values specified in the section **Scalar field**. By default, the number is set at 11.

The range between the **Min** (included) and the **Max** (included) will then be divided according to the number entered, with the following rules :

- If the specified **Min** or **Max** values lie within the range of the extreme values of the scalar field, the selected number of isosurfaces will be traced, but only for the data included in the specified range.

- If the specified **Min** or **Max** values lie beyond the extreme values of the scalar field, the isosurfaces will cover the whole set of data, but will be less than the number specified.

Method 2 :

1 - Click on **Range**.

2 - In the **Gap** edit field, enter the gap between the values marking the boundaries of an isosurface. The default value for the gap is calculated in such a way that, with the parameters by default, the result will be the same no matter which method is used.

3 - You can specify the value from which successive intervals will be calculated by adding the selected gap in the **Reference value** edit field. The reference value by default is the lowest value in the data set.

In this method, the following rules apply :

- If the specified **Min** or **Max** values lie within the range of the extreme values of the scalar field, the isosurfaces will be traced for the data included between these limit values.

- If the specified **Min** or **Max** values lie beyond the extreme values of the scalar field, the isosurfaces will cover the whole set of data.

The two methods being exclusive, the fields irrelevant to the chosen method are deactivated when selecting an option.

Display vectors

This function provides a graphic presentation of the data in a vector field. A vector layout provides broad information :

- the orientation and the direction of the vector body indicate the orientation and the direction of the field values.
- the length of the vector body may represent its module.
- the colour of vectors may be indexed on a scalar field.

Prerequisite : Prior to the display of vectors, verify the following points :

- a project must be loaded (see the documentation on project management.)
- at least one display window must already exist (see the documentation on display control.)

To specify the vector display parameters, follow these steps:

1 - After clicking on the option **Static [...]** in the **Display** menu, click on in the function toolbar at the bottom of the screen. The **Vector** dialog box will appear.

2 - In the drop-down list of the **Mesh** section, select the mesh used to generate the vector field from which you want to display vectors. This list includes all the meshes of the current project.

3 - In the drop-down list of the section **Vector field**, select the vector field from which the vectors will be traced. This list contains vector fields related to the mesh previously selected.

Note : Because of values given by default, the selected vector can be displayed now by simply clicking on **OK**.

4 - If needed, select the option **Colour** in the section **Parameters.** You will then access the **Colour parameters** page and will be able to specify the various parameters controlling colours and their indexing on a scalar field. (See *Colour parameters setup*)

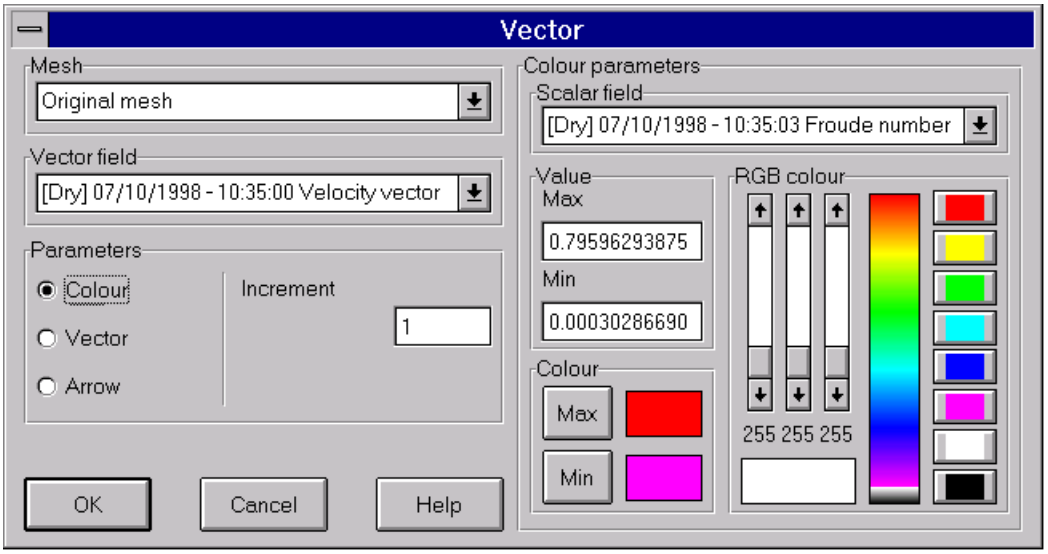
5 - In the section **Parameters**, select, if needed, the option **Vector.** You will then access the **Vector body** page and will be able to specify the format and position of the vectors. (See *Specify vector body parameters*)

6 - If needed, select the option **Arrow** in the section **Parameters.** You will then access the **Arrow** page and will be able to specify the form and dimension of the arrow point for your vectors. (See *Specify the vector point* characteristics)

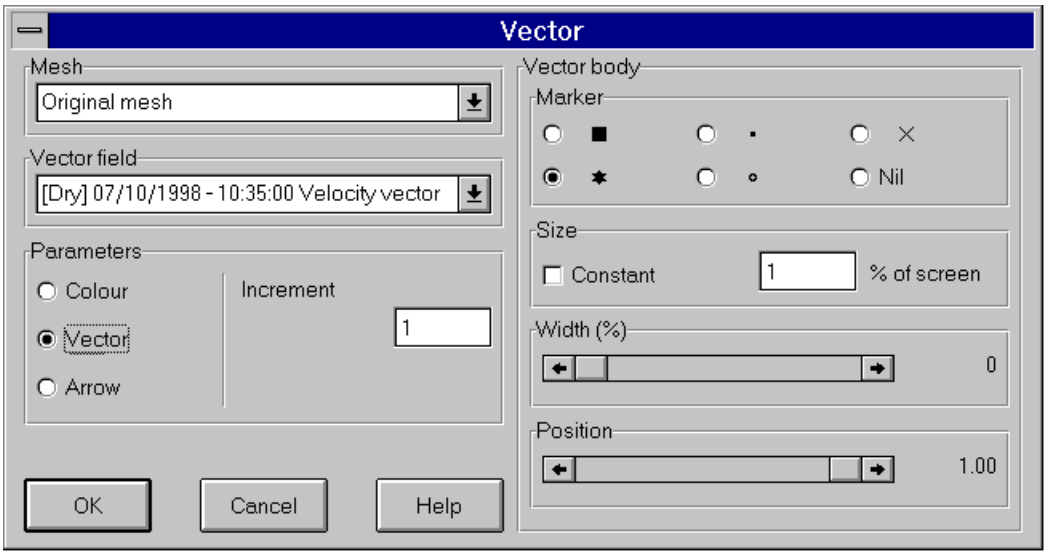
7 - In the edit field **Line increment**, specify the display density of the vectors. By default, this parameter is set at 1, to indicate that the vectors will be drawn for each mesh node.

8 - Click on **OK**.

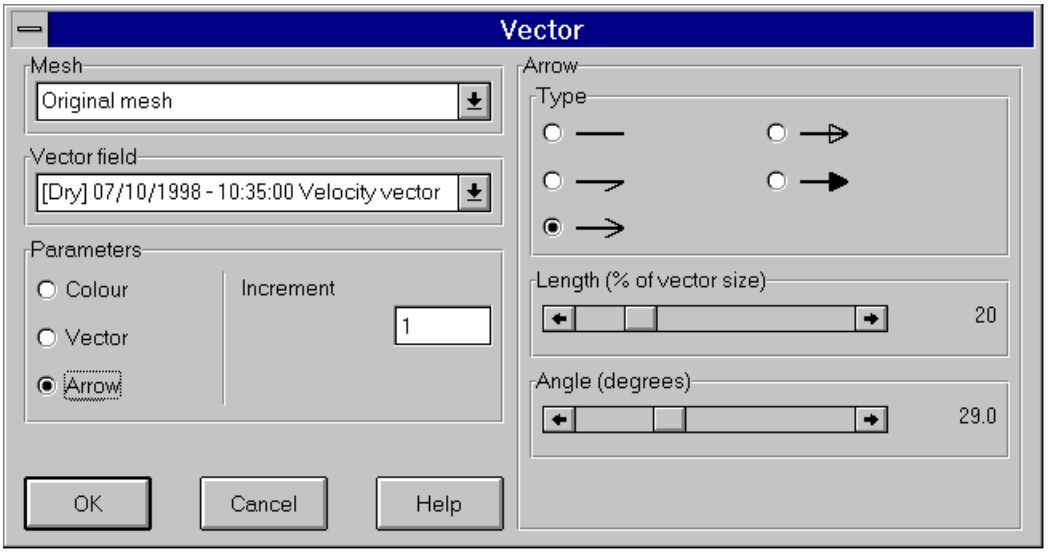
Note: If, at step 4, you have selected different colours for the min and max values, you can display the legend of the colours used. (See the section on the display of a legend in the documentation on Display control.)



Example -8 *Vectors 1/3* dialog box



Example -9 Vectors 2/3 dialog box



Example -10 Vectors 3/3 dialog box

Specify vector body parameters

The **Vector body** page of the **Vector** dialog box is used to specify the display and position parameters of the vector line :

1 - The nodes associated with vectors can be displayed under different formats selected by clicking on the appropriate option in the section **Marker**.

2 - The section **Size** offers two methods to define the display size of vectors :

- If the option **Constant** is checked off, the length of all vectors will be that of the unit vector.
- If the option **Constant** is not checked off, the display length of vectors will be a function of their module.

In both cases, the size of the unit vector in percentage of screen can be specified in the edit field.

3 - You can specify the width of the vector body as a percentage of its module by using the trackbar in the section **Width**.

4 - The position of the vector on its related node can be specified in the section **Position** : in position 1, the tail end of the vector coincides with the position of the node; in position -1, the head of the vector coincides with the position of the node; in position 0, the center of the vector coincide with the position of the node.

Specify the vector point characteristics

Within the **Vector** dialog box, the **Arrow** page is used to specify the characteristics of the vector point :

1 - You can select the desired type of arrow to represent the vectors in the section **Type**.

2 - The length of the arrow point can be adjusted on the basis of a percentage of the vector length by using the trackbar in the section **Length (% of vector size)**.

3 - The angle of the arrow point can be adjusted by using the trackbar in the section **Angle (degrees)**.

Display the mesh nodes

With this function, you can view the nodes of a mesh or of a mesh layer. Such a display provides information on the relative density of nodes. It can be enhanced by indexing the colour display of the nodes on a scalar field associated to the mesh. The numbers given to the nodes can also be displayed.

Prerequisite : Prior to the display of nodes, verify the following points :

- a project must be loaded (see the documentation on project management.)
- at least one display window must already exist (see the documentation on display control.)

To specify the node display parameters, follow these steps :

1 - After clicking on the option **Static [...]** in the **Display** menu, click on in the function toolbar at the bottom of the screen. The **Node** dialog box will appear.

2 - In the drop-down list of the section **Mesh**, choose the mesh from which you want to display the nodes. This list includes all the meshes of the current project.

Note : Because of values given by default, the selected mesh nodes can be displayed now by simply clicking on **OK**.

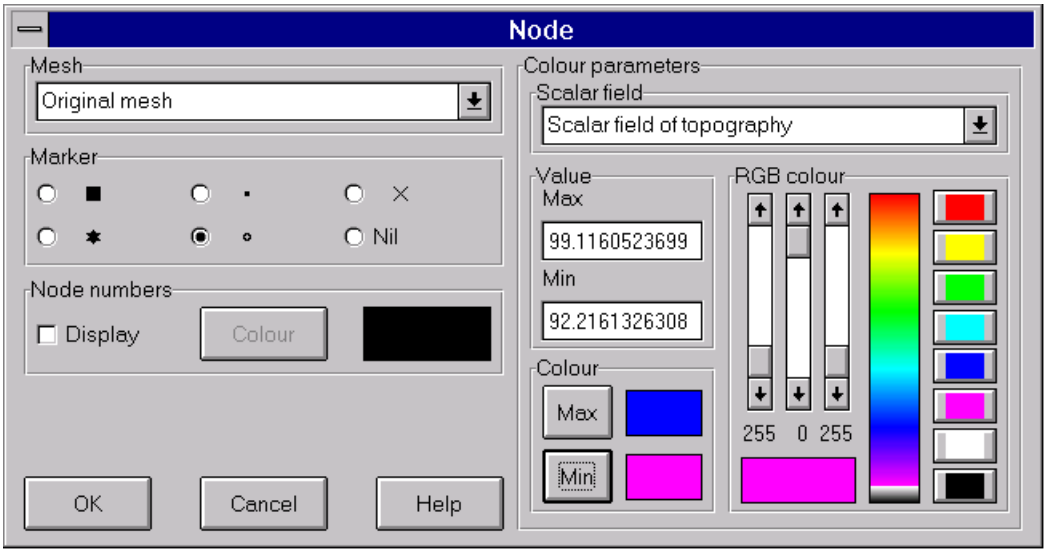
3 - In the section **Marker**, select the symbol used to represent the nodes.

4 - If wanted, modify the parameters controlling the node number display in the section **Node numbers**. (See *Display node numbers*)

5 - If needed, modify the various parameters controlling the display colours and their indexing in the section **Colour parameters**. (See *Colour parameters setup*)

6 - Click on **OK**.

Note: If, at step 5, you have elected to index the colours on a scalar field, you can display the legend of the colours used. (See the section on the display of a legend in the documentation on Display control.)



Example -11 Nodes dialog box

Display node numbers

Node numbers can also be displayed :

1 - Check off **Display** in the section **Node numbers**. The **Colour** button becomes activated.

2 - Choose the colour of numbers in the section **Colour RGB** (see *Colour parameters setup*)

3 - Click on the **Colour** button**.** The selected colour will then appear in the box associated with the button.

Display profiles

The display of profiles provides section views of selected data according to the specification of a transect. This transect must be specified in the specification mode of a profile accessible from the **Tools** menu, item **Profile**. (See documentation on ***Tools***.)

A profile specified in this mode can be further used to display a graph based on specific data. The same transect can also be used to view a section of topography, water level or the velocity module, for example.

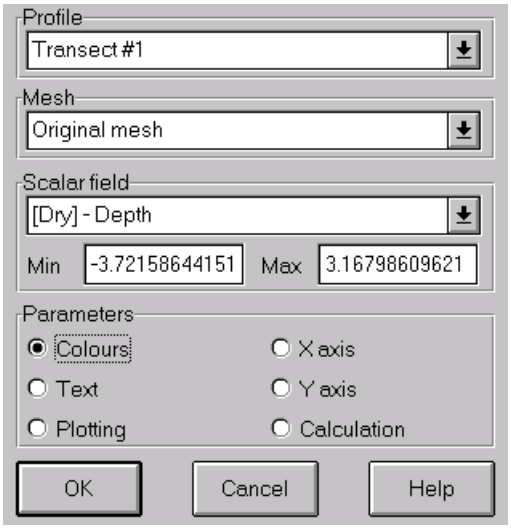
Therefore, after creating one or more profiles, you can display a profile on specific data.

Prerequisite : Prior to the display of profiles, verify the following points :

- a project must be loaded (see the documentation on project management.)
- at least one display window must already exist (see the documentation on display control.)

To specify the profiles display parameters, follow these steps ::

1 - After clicking on the option **Static [...]** in the **Display** menu, click on in the function toolbar at the bottom of the screen. The **Profile** dialog box will appear.



2 - In the drop-down list of the section **Profile**, select the desired profile. This list includes all the profiles available in your project.

3 - In the drop-down list of the section **Mesh**, choose the mesh used to generate the scalar field from which you want to display a profile. This list includes all the meshes of the current project.

4 - In the drop-down list of the section **Scalar field**, select the field for which the profile will be drawn. This list includes the scalar fields associated with the mesh previously selected. The **Min** and **Max** edit fields are assigned the maximum and minimum values of the selected scalar field.

Note : Because of values given by default, the profile can be displayed now by simply clicking on **OK**.

5 - If wanted, you can select the colours to assign to the various elements of the figure (curves, points, axis and boundaries). (See *Specify colours for profile elements*.)

6 - You can choose the text which will be displayed as title of the profile, as well as name the X and Y axes. (See *Specify profile title and axes names*.)

7 - You can select the type of line used to draw the boundaries, as well as the type of point used to mark the vertexes corresponding to the calculated values. (See *Specify types of lines and points for the profile*.)

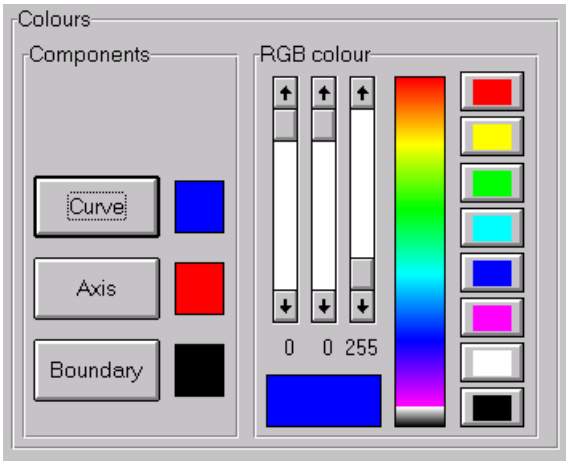
8 - You can determine the point at which the X axis must intersect the Y axis. Graduation can also be specified. (See *Specify the X axis parameters for the profile*.)

9 - You can determine the point at which the Y axis must intersect the X axis. Graduation can also be specified. (See *Specify the X axis parameters for the profile*.)

10 - Finally, you can select the calculation mode for the profile and other calculation parameters. (See *Specify the calculation parameters for the profile*.)

Specify colours for profile elements

The colour selection page provides a choice of colours which can be used to draw the profile.



The **RGB Colour** group to the right of the page works in the usual manner

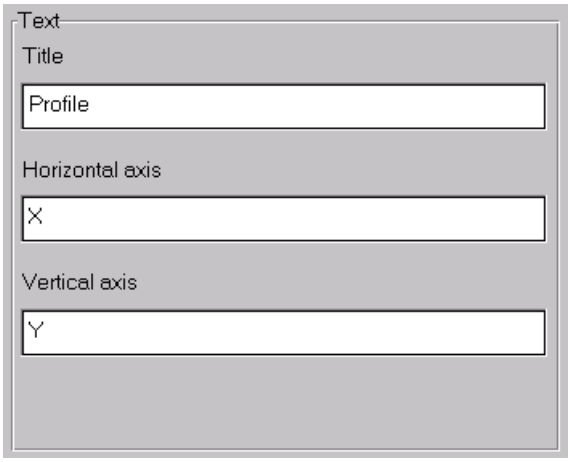
- by varying the input of the red, green and blue components using the three trackbars.
- by clicking on the desired colour in the colour spectrum bar. The *RGB* trackbars are then automatically adjusted according to the colour selected.
- by clicking on one of the predefined colour buttons. The *RGB* trackbars are then automatically adjusted according to the colour selected.

In all three cases, the colour selected will appear in the box located below the trackbars. Simply click on the appropriate button and the selected colour will be applied. The colour then appears in the box associated with the button used.

You must select colours for curves, points, axes and boundaries. Black is the default colour for all these elements.

Specify profile title and axes names

The text specification page allows to specify the profile title and the legend for the X and Y axes.



Write your text in the appropriate fields. The title will be displayed above the profile and the legends at the extremities of the axes.

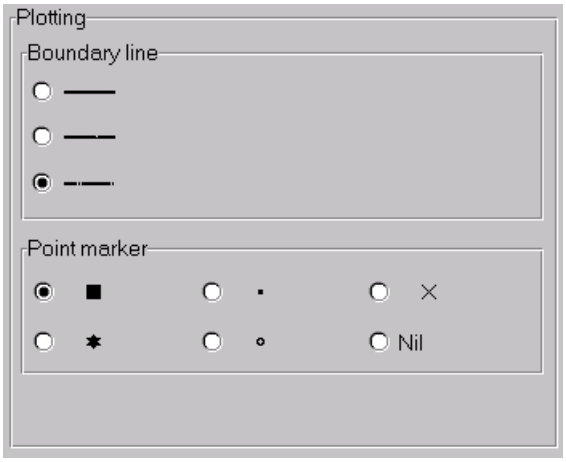
The X axis corresponds to the distance in meters from the start point of your partition of profile ; the Y axis corresponds to the value of the selected field.

Specify types of lines and points for the profile

You can choose the type of line used to draw the boundaries of the profile.

A boundary is generated at the extreme positions (minimum and maximum), and at the beginning and the end of the data. Thus, a transect may begin and/or end outside the range of the data.

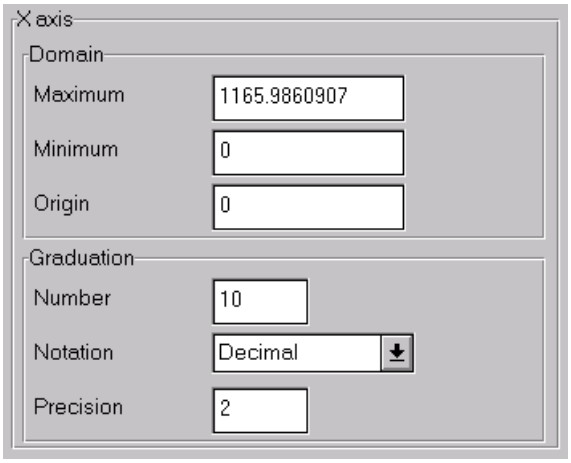
You can also specify the type of marker used to locate each value calculation point in the data field.



Click on the type of line desired (solid line, composite line, broken line). Click on the type of marker desired (square, small star, X, large star, empty circle, or no marker).

Specify the X axis parameters for the profile

The X axis parameter page specifies a number of parameters related to the X axis.



You can specify the minimum and maximum for the domain of the X axis. These correspond to the distance in meters between the start point and the end point of the partition of the profile. By default, the values correspond to the whole transect.

The point of origin selects the point at which you want the axes to intersect. By default, the axes intersect at the extreme bottom left of the profile.

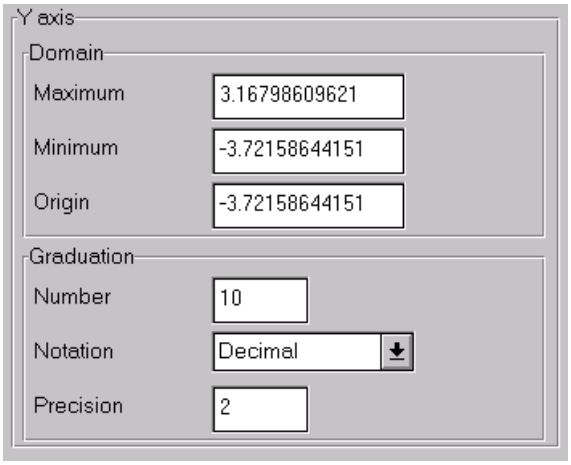
You can also specify the number of graduations to display on the X axis, along with the type of notation (decimal, scientific, engineer) and the number of decimals.

Hints :

- You can restrict the maximum to obtain round values in relation with the specified graduation. If the transect is 430.511 meters long with 10 graduations, you will obtain values with an increment factor of 43.0511. If you decreased the maximum to 400 meters long with 10 graduations, you will get an increment factor of 40 and round values on the X axis. However, you must remember that data found between 400 and 430 meters will not appear in the profile thus generated.

Specify the Y axis parameters for the profile

The Y axis parameter page specifies a number of parameters related to the Y axis.



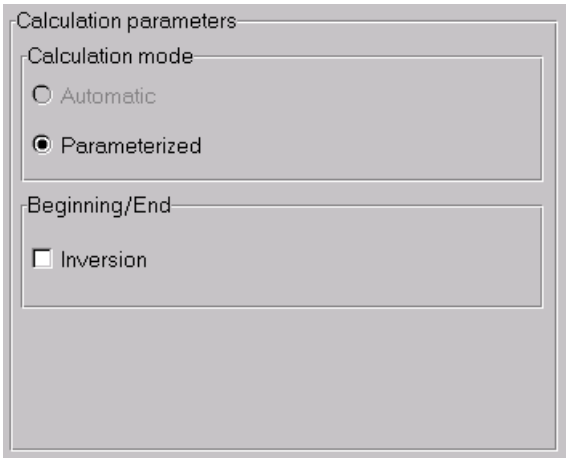
You can specify the minimum and maximum for the domain of the Y axis. These are the minimum and maximum values found in the selected scalar field. If you restrict the minimum and/or the maximum on the Y axis, and there are values exceeding your extremes, the curve will be discontinued at these points, i.e. these values will not be represented.

The point of origin selects the point at which you want the axes to intersect. By default, the axes intersect at the extreme bottom/left of the profile.

You can also specify the number of graduations to display on the Y axis, along with the type of notation (decimal, scientific, engineer) and the number of decimals.

Specify the calculation parameters for the profile

The calculation parameter page defines the final parameters needed to draw the profile.



You can specify the calculation mode used.

Currently, only the parameterized calculation mode is available. This method to calculate the position of the points to evaluate is controlled by the information found in the partition of the profile. In this partition, you have specified for each vertex of your

transect the distance between the values (5 meters, 10 meters, etc.).

The automatic calculation mode will not consider the information found in the partition of the profile but will rather take into account the data support mesh. Each time a segment of the mesh will be crossed along the path of the transect, the value will be assessed. The final result will depend on the intricacy and density of the mesh.

The beginning of the transect can also be reversed. By default, the evaluation is conducted from the beginning to the end. By reversing, the evaluation would be made from the end of the transect toward the beginning.

Cartographic dressing

The software offers a range of cartographic dressing options to complete, document and explain a map of results.

Prerequisite : Before setting a cartographic dressing, the following conditions must be met :

- a project must be loaded (see the documentation on project management.)
- at least one display window must already exist (see the documentation on display control.)

To access these functions, you must be in the cartographic dressing mode available in the **Display** menu, item **Cartographic dressing [...]**.

There are two categories of functions in the cartographic dressing options :

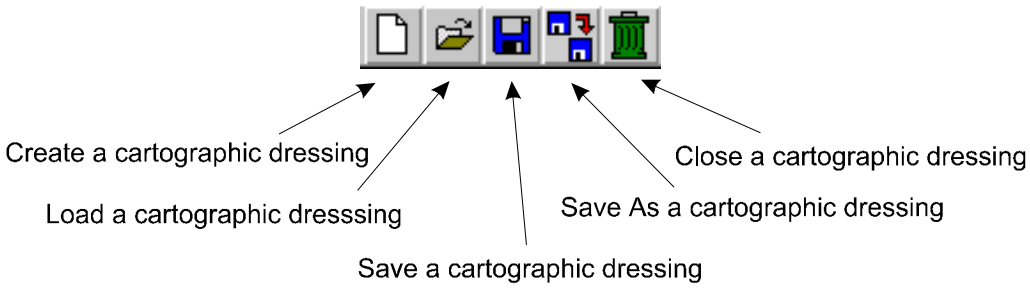
1 - Cartographic dressing management

2 - Element management.

Cartographic dressing management

A series of function buttons is displayed at the bottom of the screen when the cartographic dressing mode is accessed.

Figure 2 Cartographic dressing management tool bar


Create a cartographic dressing
Load a cartographic dresssing
Save a cartographic dressing
Save As a cartographic dressing
Close a cartographic dressing

The cartographic dressing mode allows to create, load, save, save as and close a cartographic dressing. The cartographic dressing comprises several cartographic dressing elements.

The cartographic dressing management options are :

- Create a cartographic dressing
- Load a cartographic dressing
- Save a cartographic dressing
- Save As...a cartographic dressing
- Close a cartographic dressing

To add cartographic dressing elements, you must first create or load an existing cartographic dressing.

For information on cartographic dressing element management, see section Element management.

Create a cartographic dressing

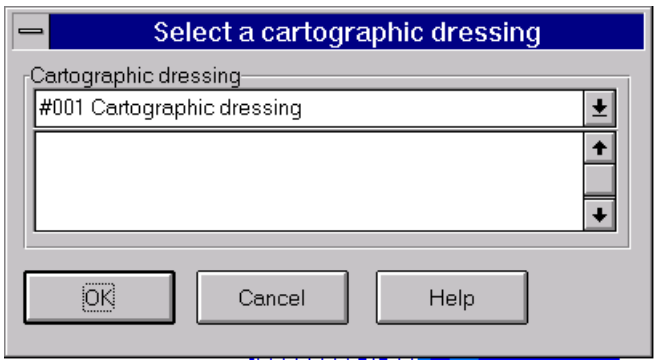
To create a cartographic dressing, click on the function button in the cartographic dressing management function tool bar.

If a cartographic dressing had already been set, you will be asked to save it ; it will then be closed and replaced by the new cartographic dressing.

To add cartographic dressing elements, see section Element management.

Load a cartographic dressing

To load a cartographic dressing, click on the function button in the cartographic dressing management function tool bar. A cartographic dressing selection dialog box will appear.



In the drop-down list, select the desired cartographic dressing. Click on **OK** to load the cartographic dressing or **Cancel** to close the operation.

If a cartographic dressing had already been set, you will be asked to save it ; it will then be closed and replaced by the cartographic dressing you wish to load.

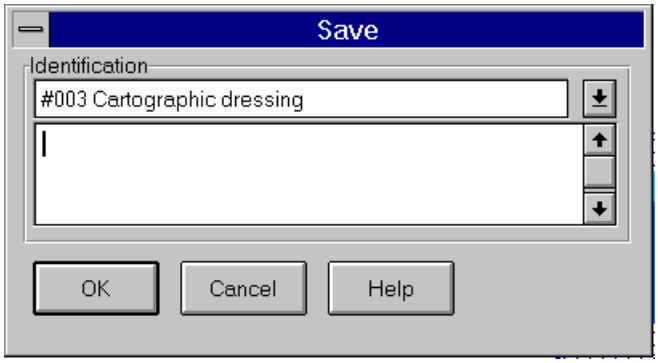
To manage cartographic dressing elements, see section Element management

Save a cartographic dressing

To save a cartographic dressing, click on the function button in the cartographic dressing management function tool bar.

If the cartographic dressing has already been saved, the recording is immediate. An hour-glass appears on the screen ; when it disappears, recording is completed.

If the cartographic dressing has not previously been saved, you will be asked to provide a label and a description for the new cartographic dressing.



Type the label in the upper part of the **Identification** group. You can draw from existing labels for similar data. However you must provide a label that is unique.

Type a description in the lower part of the **Identification** group.

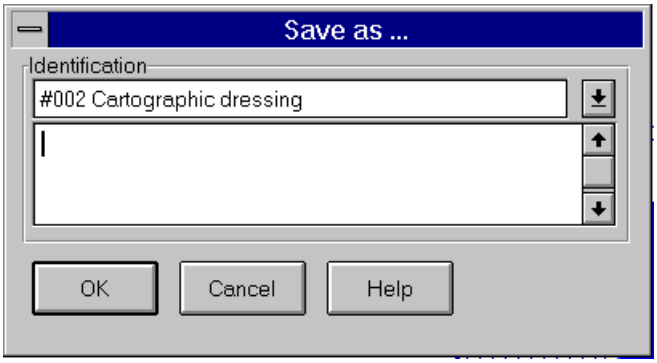
Click on **OK** to confirm the operation or on **Close** if you do not want to save the cartographic dressing.

Save As...a cartographic dressing

The cartographic dressing Save As... option takes a complete copy of the current cartographic dressing. To save as..., click on the function button in the cartographic dressing management function tool bar.

If the cartographic dressing has not previously been saved, the procedure will be the same as for the save option. (See *Save a cartographic dressing*.)

However, if the cartographic dressing has already been saved, the Save As... dialog box appears.



You must type a unique label for this type of data in the upper part of the **Identification** group. The label displayed is that of the cartographic dressing to be saved as... ; it cannot be used as is.

You can draw from it or from any other label to write a new unique label.

Type a description in the lower part of the **Identification** group.

Click on **OK** to confirm the Save As... operation or on **Cancel** if you do not want to Save As... . The current cartographic dressing becomes the cartographic dressing saved as... .

Close a cartographic dressing

If you wish to remove a cartographic dressing from the display, you must absolutely follow this procedure. Click on the function button in the cartographic dressing management tool bar.

If the entity has been modified, a pop-up window appears to show the current status.

Click on **Yes** to save, **No** to delete the modifications and **Cancel** to cancel the closure operation.

Notes :

- This operation does not delete the cartographic dressing ; it simply removes it from the display. Since this entity is not static, as you can interact with its elements, it cannot be closed in the **Display list management** window. (See documentation on *Controlling the display*.)
- To delete a cartographic dressing from the database, you must go to the **Entity manager** in **Project management**. (See documentation on *Project management*.)

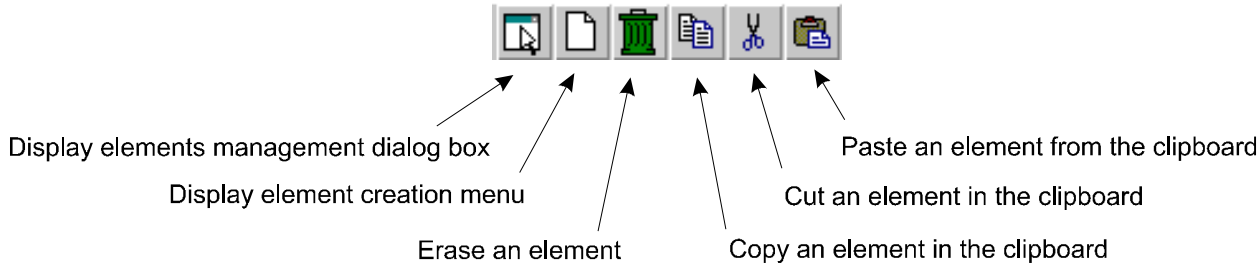
Element management

A series of function buttons are available to manage the dressing elements within a given cartographic dressing.

Prerequisite :

- To manage the cartographic dressing elements, you must have created or loaded a cartographic dressing. (See *Cartographic dressing management*.)

Figure 3 Element edit functions tool bar

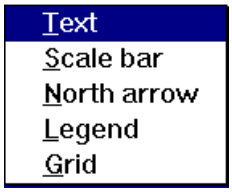

Display elements management dialog box
Display element creation menu
Erase an element
Copy an element in the clipboard
Cut an element in the clipboard
Paste an element from the clipboard

The management of the cartographic dressing elements can only be done inside a cartographic dressing. The management function tool bar allows to :

- Create an element
- Modify an element
- Delete an element
- Copy an element
- Cut an element
- Paste an element

Create an element

To create a cartographic dressing element, click with the right mouse button on the function button in the cartographic dressing element management tool bar. A pop-up menu appears.



In this menu, select the type of element you want to create. The cartographic dressing element is immediately displayed on the screen.

To modify the element, you must open the pop-up window **Management of cartographic dressing elements** by clicking on the function button in the cartographic dressing element management tool bar. (See section *Modify an element*.)

Notes :

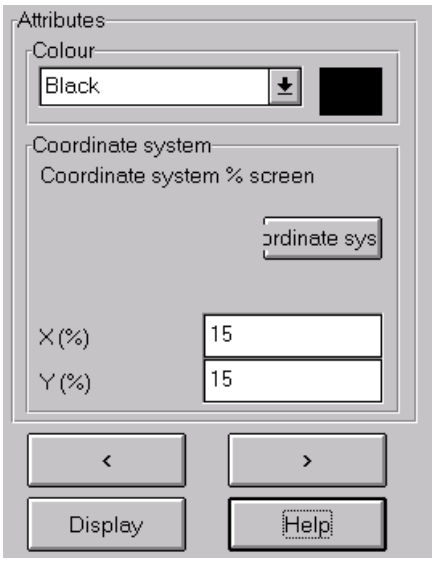
- In the specific case of a legend, you must have previously saved a graphic element such as an isoline or an isosurface to create a legend on this element. (See documentation on *Controlling the display*, section *Saving a graphic element*.)

Modify an element

To modify a cartographic dressing element, you must open the Cartographic dressing element management pop-up window by clicking on the function button in the cartographic dressing element management tool bar. The window is displayed on the screen.

This window always displays the active cartographic dressing element. To activate a cartographic dressing element, click directly on this element on the screen, or using the navigation

arrows in the Cartographic dressing element management window, move to the appropriate element.



This is a representation of the left part of the window which is common to all types of cartographic dressing elements.

With this window, you can do the following operations :

- Assign a position to an element
- Assign a colour to an element
- Modify the parameters of the text
- Modify the parameters of the scale bar
- Modify the parameters of the North arrow
- Modify the parameters of the legend
- Modify the parameters of the grid

Notes :

- You must click on **Apply** for the modifications to be applied.

Assign a position to an element

It is possible to assign a position in the coordinate system of your choice.

For the cartographic dressing, you have access to a special coordinate system, the percentage of screen. Positions in X and Y represent the position in percentage from the lower left corner.

The other coordinate systems correspond to what is available in your project. Thus, if your project is georeferenced, you should have at least one cartographic coordinate system (UTM, MTM, ...) and one geographic coordinate system (long/lat). If your

project is not georeferenced, you should have a cartesian coordinate system.

There are two methods to assign a position to a cartographic dressing element :

1 - Move the element directly on the screen

2 - Specify the position explicitly in the Cartographic dressing element management window.

In the first method, click on the element to move and use the drag and drop technique to reach the desired position. You can still choose the coordinate system in which you want it to be assigned.

In the second method, you must choose the type of coordinate system and specify the X and Y coordinates in the selected system.

Notes :

- You must click on **Apply** for the modification to be applied.

Assign a colour to an element

All cartographic dressing elements can be displayed in a given colour. To select a colour for an element, follow these steps :

1 - Select the desired element by clicking directly on it or by using the navigation tools of the Cartographic dressing element management window.

2 - In the drop-down list in the left portion of the window, select a colour.

3 - Click on **Apply** for the modification to be applied.

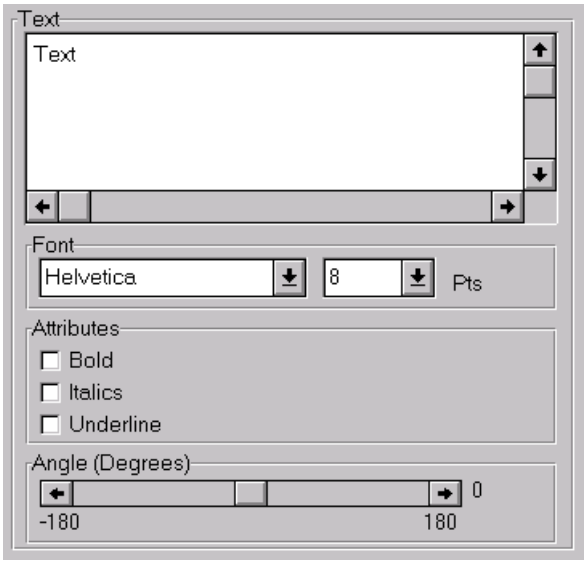
Warning :

- White is available. However, if your screen background is white, the element will disappear when it is not selected anymore.
- Red is available. However, this colour is used to represent the selected element. If you choose this colour, your element will appear as if it is always selected.

Modify the parameters of the text

You must first select a cartographic dressing element of the type **Text**.

The right hand page of the Cartographic dressing element management window gives you the possibility to specify the parameters of the text.



In the first edit field, you can specify the text to display.

You can modify the font by selecting the desired font in the drop-down list **Font**. The second drop-down list in this group is to select the font size.

You can modify the font specifications by clicking on the check boxes to this purpose. Available to you are bold typeface, italics and underlined typeface. These options are not mutually exclusive.

Finally, you can rotate your text by clicking on the trackbar. The rotation angle appears at the right of the bar.

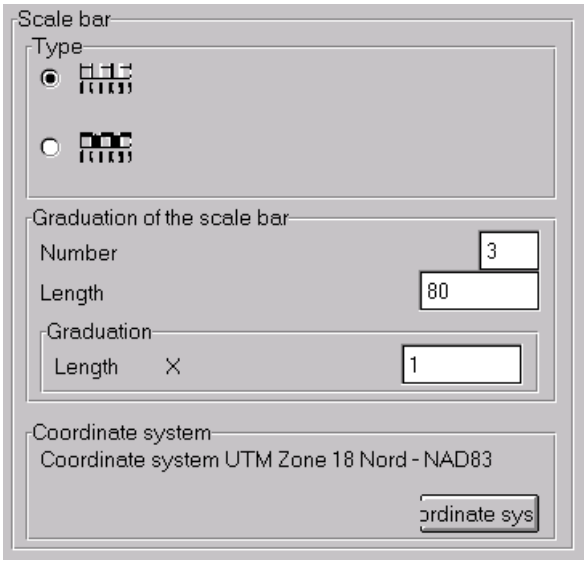
Notes :

- You must click on **Apply** for the modification to be applied.
- Text on more than one line is not possible for the moment.
- The rotation angle works only with the fonts Helvetica and Times.

Modify the parameters of the scale bar

You must first select a cartographic dressing element of the type **Scale bar**.

The right hand page of the Cartographic dressing element management window gives you the possibility to specify the parameters of the scale bar.



You can display two types of scale bar, one without filling and one with filling. Select a type by clicking on the option desired.

You can select the coordinate system in which the bar will display its graduations. In the case of a georeferenced project, the basic unit is the meter and you will have the use of all the cartographic coordinate systems of the project. If your project is not georeferenced, the basic unit is the abstract unit of the project and you will have the use of all the cartesian coordinate systems of the project.

Finally, the graduation and spacing of the scale bar can be specified. You can choose the number of intervals displayed, 3 being the default number. You can specify the length corresponding to an interval. For example, in the case of a georeferenced project, 80 meters per interval. Finally you can give a multiplication factor for the graduation. This option is not cohesive with data management and should remain at 1.

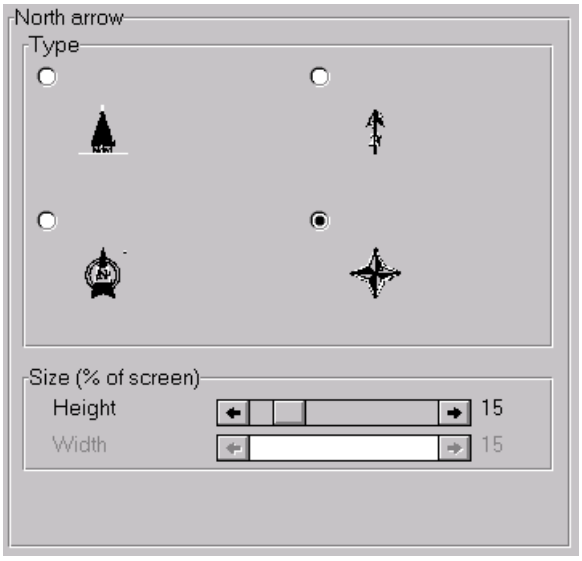
Notes :

- You must click on **Apply** for the modification to be applied.

Modify the parameters of the North arrow

You must first select a cartographic dressing element of the type **North arrow**.

The right hand page of the Cartographic dressing element management window gives you the possibility to specify the parameters of the North arrow.



You have the choice of four types of north arrow : a triangle, a thin arrow, an arrow with a circle and an arrow with four points. Click on the desired option.

Next, you can specify the percentage of screen the arrow will occupy. The height can be specified for all the types of arrow. The width can also be specified for the first type of arrow.

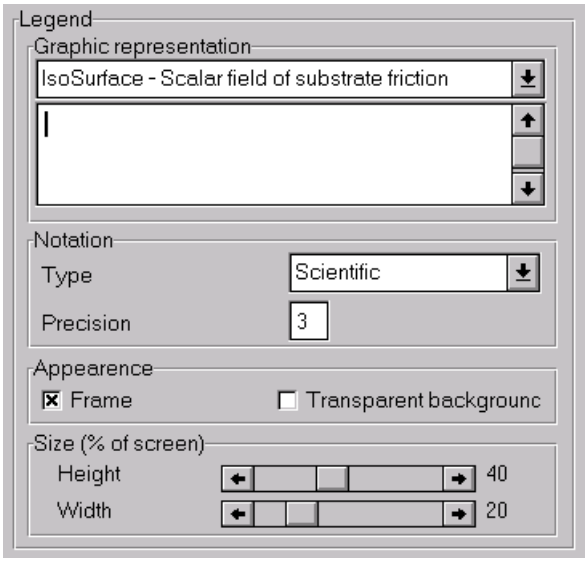
Notes :

- You must click on **Apply** for the modification to be applied.

Modify the parameters of the legend

You must first select a cartographic dressing element of the type **Legend**.

The right hand page of the Cartographic dressing element management window gives you the possibility to specify the parameters of the legend.



You must select, in the drop-down list, a graphic element on which you want to display a legend.

You can choose the type of numerical display in the legend : scientific, decimal or engineer. Make your selection in the drop-down list of the group **Notation**. You can also specify the precision of these values, i.e. the number of decimals.

You can also specify the appearance of the legend. You can have a frame and a transparent background, so that you can still see what is behind.

Finally, the size of the legend in height and width can be specified by using the appropriate trackbar. The specified percentage is displayed at the right of these bars.

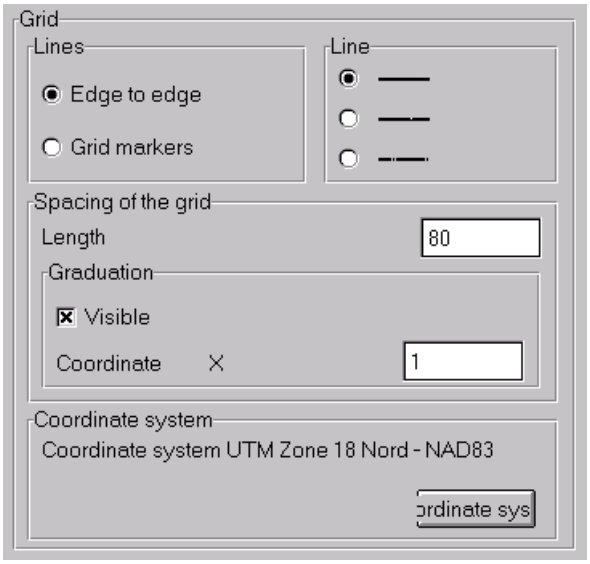
Notes :

- In the specific case of a legend, you must have previously saved a graphic element such as an isoline or an isosurface to create a legend on this element. (See documentation on *Controlling the display*, section *Saving a graphic element*.)
- You must click on **Apply** for the modification to be applied.

Modify the parameters of the grid

You must first select a cartographic dressing element of the type **Grid**.

The right hand page of the Cartographic dressing element management window gives you the possibility to specify the parameters of the grid.



You can choose a grid covering the whole window, from left to right and from top to bottom, or simply grid markers around the window. Click on the desired option.

You can select the type of line : solid line, broken line (dot-dash-dot) or dash-dash line. Click on the desired option.

You can select the coordinate system in which the grid will display its graduations. In the case of a georeferenced project, the basic unit is the meter and you will have the use of all the cartographic coordinate systems of the project. If your project is not georeferenced, the basic unit is the abstract unit of the project and you will have the use of all the cartesian coordinate systems of the project.

Finally, the graduation and spacing of the grid can be specified. You can specify the length corresponding to an interval. For example, in the case of a georeferenced project, 80 meters per interval. You can display or not the numerical values associated with each graduation. Finally you can give a multiplication factor for the graduation. This option is not cohesive with data management and should remain at 1.

Notes :

- You must click on **Apply** for the modification to be applied.

Delete an element

To delete a cartographic dressing element, follow these steps :

1 - Select the element to be deleted by clicking directly on it or by using the navigation tools of the Cartographic dressing element management window.

2 - Click on the function button in the cartographic dressing element edit tool bar.

3 - The element is deleted from the cartographic dressing.

Copy an element

To copy a cartographic dressing element, follow these steps :

1 - Select the element to be copied by clicking directly on it or by using the navigation tools of the Cartographic dressing element management window.

2 - Click on the function button in the cartographic dressing element edit tool bar.

3 - The element is copied in the clipboard.

You can then paste it as you wish. (See *Paste an element*.)

Cut an element

To cut a cartographic dressing element, follow these steps :

1 - Select the element to be cut by clicking directly on it or by using the navigation tools of the Cartographic dressing element management window.

2 - Click on the function button in the Cartographic dressing element edit tool bar.

3 - The element is copied in the clipboard and removed from the cartographic dressing.

You can then paste it as you wish. (See *Paste an element*.)

Paste an element

To paste a cartographic dressing element, you must have previously copied or cut a cartographic dressing element. (See *Copy an element* and *Cut an element*.)

To paste a cartographic dressing element, follow these steps :

1 - Click on the function button in the cartographic dressing element edit tool bar.

2 - The element will be copied slightly shifted from the original copy.

Chapter 6 Data preparation

The preparation of your data is an essential step in the process of constructing the numerical terrain model. After importing the terrain data in the database of the project, you must integrate them to the model and complement them with other data reflecting the reality to be modelized.

The numerical terrain model is a homogenous structure in which heterogeneous data are integrated. In MODELEUR, the integration is conducted through a graphic structure called **Partition** which splits the study domain in sub-domains on which specific parameters can be defined. The different types of data are specified on partitions, to be transferred on a finite element mesh which has been generated from a partition.

In order for you to prepare your data in a controlled and efficient manner, MODELEUR provides the following functions :

- Generate a mesh
- Projection of the topography
- Specify the boundary conditions
- Specify the wind characteristics
- Specify the substrate characteristics
- Specify the ice characteristics
- Specify the other friction coefficients

These functions are represented by software modules called **Modes**. Each mode presents a function button tool bar at the bottom of the screen. These buttons are divided in various groups according to their use. In addition to the group of buttons used to control the display, the modes have two other groups of buttons in common allowing to manage and edit the partitions related to the type of data to be processed. These are described in the section *Specification tool : the partition*.

Notes :

- Although the text seems to impose a logical and unchanging order, these tasks can be and will often have to be conducted with frequent look back to correct several details. Several branches can be conducted in parallel to test several alternatives.

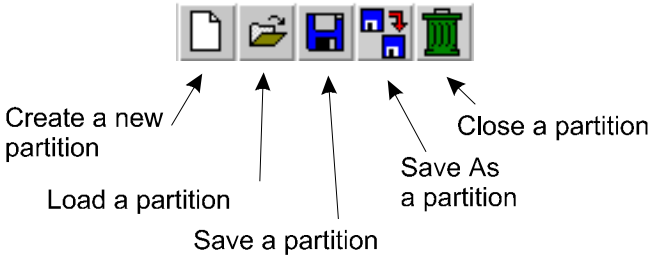
Specification tool : the partition

The partition is a graphic structure allowing you to specify data on separate areas of the study domain. For each type of data, there is a particular type of partition (e. g. : substrate partition, topography partition, etc...).

The partitions are represented by entities in the database, and each mode provides the necessary functions to manage the partition entities it creates :

- Create a new partition
- Load an existing partition
- Save a partition
- Save a partition under a different name
- Close a partition

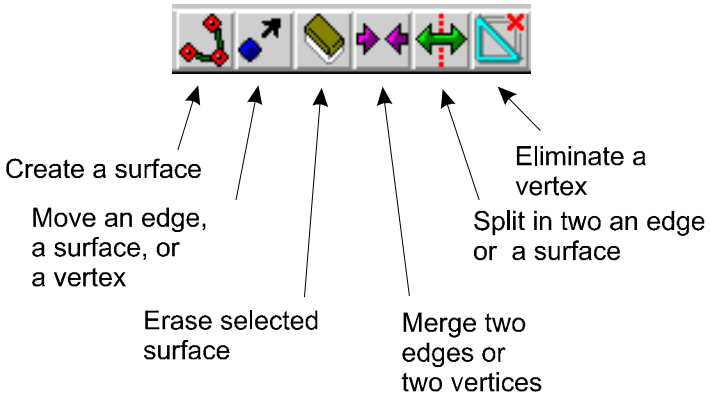
Figure 4 Group of partition management buttons



A partition is made of a series of surfaces splitting the study domain in as many sub-domains. A partition is built in an interactive manner, i. e. by « drawing » one by one each sub-domain. In each mode, a group of function buttons gives you access to the functions necessary to draw and handle the sub-domains. These functions are :

- Create a sub-domain
- Move an element of a partition
- Erase a sub-domain from a partition
- Merge two elements of a partition
- Split an element of a partition
- Delete a vertex

Figure 5 Group of partition edit buttons


Create a surface
Move an edge, a surface, or a vertex
Erase selected surface
Merge two edges or two vertices
Split in two an edge or a surface
Eliminate a vertex

Create a new partition

The partitions are part of the database as distinct entities. Therefore, before you can graphically construct a partition (see *Create a sub-domain*), you must create an entity for it in the database of the project. To do this, each mode has the function button at the left end of its tool bar. When this button is activated, a partition entity of the specific type of the current mode is created in the database of the project.

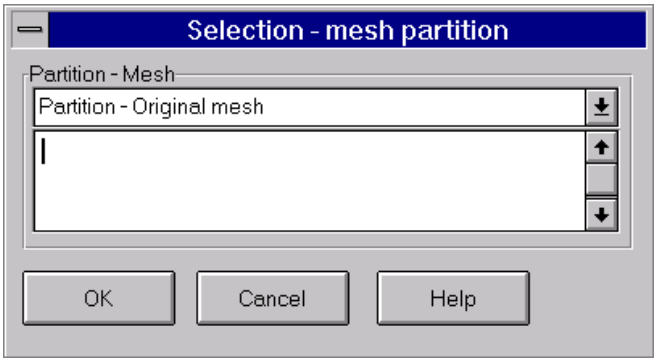
Notes :

- This button remains accessible during the whole work session in the current mode, so that you can create a new partition at any time. However, since only one partition can be handled at a time, if you attempt to create a new partition while another one is already opened, you will have to close the latter. The closure is effected automatically by the software. If the partition to be closed has been modified, a dialog box will inform you of the situation and you can save the partition before it is closed.
- The software automatically gives a unique name to the new partition created. You can modify this name and, if you wish, add a description by doing an explicit save operation. (See *Save a partition*)
- Certain partitions require additional information to create a partition. Special cases will be explained in the documentation treating of each partition.

Load an existing partition

A partition is part of the database of the project and can be loaded at will in the mode used to construct it :

1 - Activate the function button located in the left part of tool bar of the current mode. The **Selection of a partition** dialog box appears. It contains a drop-down list of all the partitions specific to the current mode.



2 - In the drop-down list, select the partition that you wish to load. Its description is displayed in the lower part of the dialog box.

3 - Click on **OK** to load that partition or on **Cancel** if you do not want to load it.

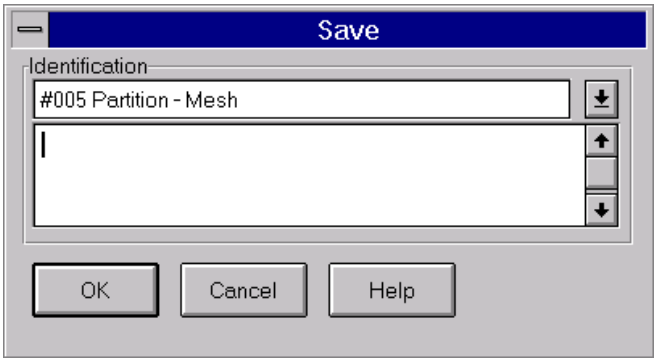
Note :

- Only one partition can be handled at a time. If a partition is already loaded, it must be closed before the selected partition can be loaded. The closure is effected automatically by the software. If the partition to be closed has been modified, a dialog box will inform you of the situation and you can save the partition before it is closed.

Save a partition

You can save the partition you are working with at any time. Simply, activate the function button common to all the modes in the tool bar at the bottom of the screen.

If it is a newly created partition (see *Create a new partition*), the **Save** dialog box appears and you can modify the default name given to the partition by the software when it was created, and if wanted, add a description.



1 - In the upper part of the section **Identification**, you can modify the default name given by the software. The drop-down list provides the names of existing partitions from which you can draw inspiration. However, the data model requires that the name of the partition must be unique among the partitions of the same type. A warning message will inform you if you do not respect this rule.

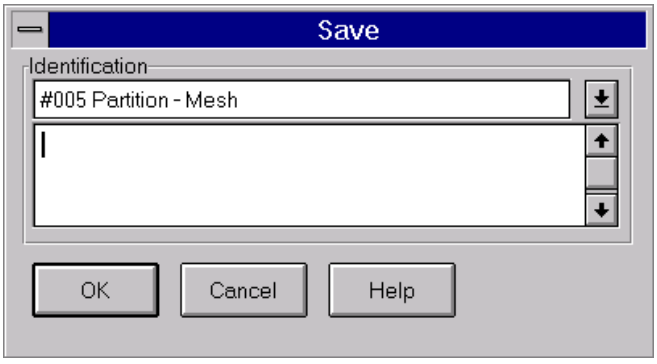
2 - In the lower part of the section **Identification**, enter, if wanted, a description of the partition. There are no restrictions here.

3 - Click on **OK** to save the partition or on **Cancel** to cancel the operation.

Save a partition under a different name (Save As)

Copies of a same partition can be created at will in the database :

1 - Activate the function button in the tool bar of the current mode at the bottom of the screen. The **Save As...** dialog box will appear.



2 - In the upper part of the section **Identification**, enter the name that you want to give to the copy of the partition. The drop-down list provides the names of existing partitions from which you can draw inspiration. However, the data model requires that the name of the partition must be unique among

the partitions of the same type. A warning message will inform you if you do not respect this rule.

3 - In the lower part of the section **Identification**, enter, if wanted, a description of the partition.

4 - Click on **OK**. A copy of the current partition in its present state is created in the database and it automatically becomes the current partition.

Note :

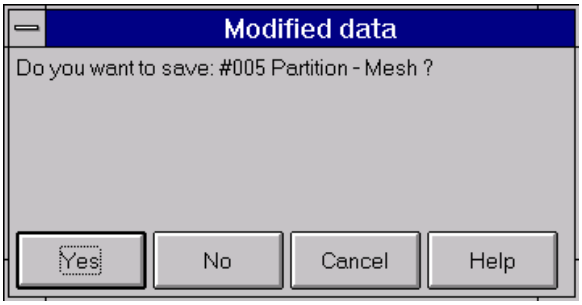
- The original partition is closed without being saved ; this implies that any changes effected since the last explicit save operation (see *Save a partition*) will be retained only in the new copy.

Close a partition

Once a partition is loaded or has just been created, it remains in the system memory until it is closed. Therefore, it remains displayed in the display windows where it is loaded. To remove a partition from the display and, if wanted, to save it in its current state, you must close it :

1 - Activate the function button in the tool bar of the current mode at the bottom of the screen.

2 - If the partition has been modified, a dialog box will warn you and ask you if you want to save it before closing it.



Click on **Yes** to save, on **No** if you do not want to save the changes, and on **Cancel** to cancel the closing of the partition.

Notes :

- This operation does not delete the partition, it simply removes it from the display. Since this entity is not a static entity because you can interact with its elements, it cannot be closed in the **Display list management** window. (See the documentation on *Controlling the display*.)

- To delete a partition from the database, you must go to the **Entity manager** of the **Projects** menu. (See the documentation on *Project management*.)

Create a sub-domain

To create a sub-domain, follow these steps :

1 - Activate the function button in the edition tool bar of the current mode. With the exception of the view control buttons, all the other buttons become inactive.

2 - In the active window, click successively on the positions of the vertexes of the surface to be created. At each click, a vertex appears as a circle and a edge links it to the preceding vertex. If a position does not respect the validity constraints of the partition, the vertex is not created and a sound signal is heard.

3 - Close the surface by clicking again on the first vertex created. All the buttons become active again.

Note :

- During step 2, if you click again on the function button, you erase the entire sub-domain being created.
- Since the information is carried by the partition on the vertexes, the edges or the surfaces, default parameters are imposed during the creation. It may be useful to specify the default parameters **before** beginning the construction. See the specific section on the parameters of each partition.

Hint :

- To position the vertexes with more precision, use the view control buttons to zoom in and re-center the image on the target zone. During the entire process of sub-domain interactive construction, the view control buttons remain active. (See the documentation on *Controlling the display*).

Move an element of a partition

A partition is made of sub-domains which are themselves made of vertexes and the edges that link them. Each element of the partition can be moved at will, as long as the validity constraints of the partition are respected. Whether you are moving a sub-domain, an edge or a vertex, the principle is the same :

1 - Activate the function button in the edition tool bar of the current mode. With the exception of the view control buttons, all the other buttons become inactive.

2 - In the target partition, select the element to be moved by clicking with the left mouse button.

3 - Maintain the left button pressed down and move the element to the desired position ; then release the button (drag and drop operation).

4 - Repeat steps 1 to 3 to make all the adjustments needed.

5 - Click again on the function button of step 1. All the buttons become active again.

Note :

- You can move a vertex, an edge or a surface of the partition by selecting the appropriate element.

Hint :

- To move an element over a long distance with more precision, use the view control buttons (See the documentation on *Controlling the display*):
  a) Make a zoom out.
  b) Move the element to its approximate position.
  c) Zoom in on the zone of the new position.
  d) Adjust the position with more precision.
  e) Return to the initial view.

Erase a sub-domain from a partition

To erase a sub-domain from a partition, follow these steps :

1 - Click on the sub-domain to be erased with the left mouse button.

2 - Activate the function button .

Note :

- It is possible to erase several sub-domains in a single operation. During step 1, select an element (vertex or edge) that is common to the sub-domains you want to erase. However, on complex partitions, this technique may produce results hard to foresee.

Merge two elements of a partition

Two vertexes or two edges belonging to two sub-domains can be merged into a single element. The two sub-domains are then fused together by the common vertex or edge :

1 - Click with the left mouse button on the vertex or the edge to be merged in one of the target sub-domains.

2 - In the other sub-domain, select in the same manner the vertex or the edge to merge, but this time, maintain the **Ctrl** key of the keyboard pressed down. The two elements selected will appear in red.

3 - Activate the function button . If the merge operation respects the validity constraints of the partition, it will be effected ; otherwise, a sound signal will be heard.

Note :

- The merge operation always proceeds by a « displacement » of the second element selected toward the first one.

Split an element of a partition

It is possible to split an edge or a sub-domain. In the case of an edge, it will be divided in two by the addition of a new vertex in its middle. In the case of a sub-domain, an edge linking two of its vertexes is added, thus creating two new sub-domains from the original sub-domain.

To split an edge :

1 - Click on the edge to be split with the left mouse button.

2 - Activate the function button .

To split a sub-domain :

1 - Press down the **Ctrl** key of the keyboard, click with the left mouse button on the two vertexes between which the edge, common to the two new sub-domains resulting from the split, will be drawn.

2 - Activate the function button . If the split operation respects the validity constraints of the partition, it will be effected ; otherwise, a sound signal will be heard.

Note :

- Any edge can be split. However, only the sub-domains with a minimum of four vertexes can be split.

Delete a vertex

Deleting one of the vertexes of a sub-domain has the following result : the two edges linking the deleted vertex to the

neighboring vertexes are replaced by a single edge linking the latter. This operation can only be conducted if the target vertex is isolated, i. e. it belongs to only one sub-domain. Also, the sub-domain involved must initially have at least four vertexes so that the resulting surface is not nil.

To delete a vertex, follow these steps :

1 - Click on the vertex to be deleted.

2 - Activate the function button at the bottom of the screen. If the above-mentioned conditions are respected, the vertex is deleted ; otherwise, a sound signal will be heard.

Generate a mesh

The mesh generation mode provides an extremely powerful tool to generate meshes.

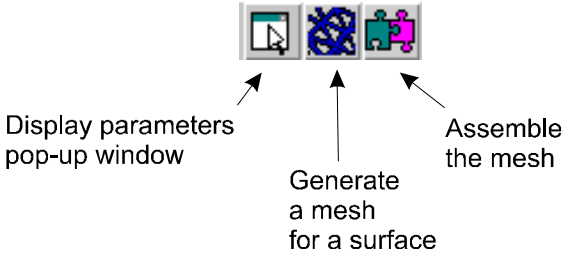
You can interactively build a mesh partition and specify the generation parameters. You can generate the mesh and inspect the result ; you can start over at will. When you are satisfied with the mesh generated, you assemble it to conserve it as a full time mesh.

The strength of this tool is that you do not have to place each node manually in the mesh. Indeed, to build a mesh with 25 nodes does not represent a problem ; to build a mesh with 200 nodes begins to be a struggle. However, to build a mesh with 4000 nodes is almost impossible without a conception aid tool.

To understand the principle of the modeling tool that is the partition, see the section *Specification tool : the partition.* You will learn how to construct, edit, and manage partitions.

In this section, we look at the specific aspects of mesh partitions. You have access to buttons specific to the mesh partition, known as the mesh partition control buttons.

Figure 6 Group of mesh partition control buttons



- Specify the mesh parameters
- Generate a mesh on a surface
- Assemble the global mesh.

Specify the mesh parameters

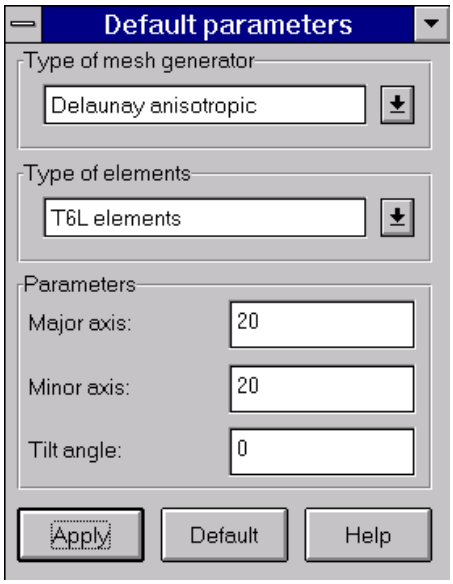
In a mesh partition, parameters must be specified to describe the type of mesh you want to generate, the generation algorithm you want to use, and finally the spacing criteria to apply between the nodes of the mesh to be generated.

Consult the following sections :

- Global parameters
- Default parameters
- Parameters specific to a vertex .

Global parameters

Global parameters apply to the partition as a whole. Click on the function button in the mesh partition control tool bar. The mesh parameters pop-up window will appear. To access the global parameters specification, you must not select any element of the partition. The title of the window should be **Default parameters**.



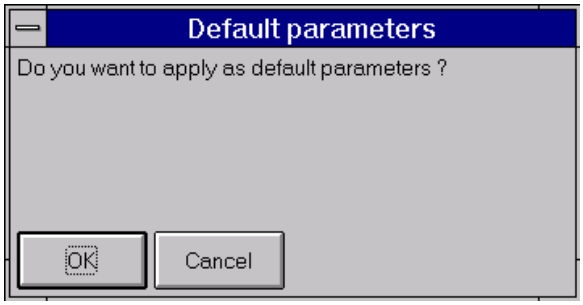
The global parameters applicable on a mesh partition are the type of mesh element and the generation algorithm used :

- Type of mesh element
- Mesh algorithm generation

Note :

- You can display this window by double-clicking in the display window.

For your selection to be effective, you must click on the command button **Apply** in the parameter window. A confirmation box is displayed.



Click on **OK** to apply or on **Cancel** if you do not want to apply the parameters.

Type of mesh element

You must select the type of mesh elements in the group **Type of elements** in the parameters window. In the drop-down list, you can choose between the T3, T6L and T6 elements. The T6L is the element by default.

Warning :

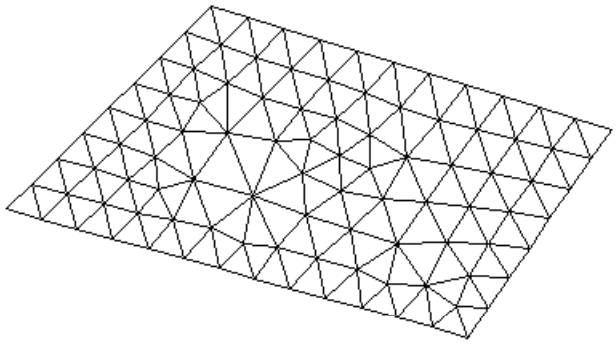
- Although three types of mesh elements are available, only the **T6L** elements are appropriate for hydrodynamics simulation. If you generate a mesh to be used in a simulation, you must absolutely use the **T6L** element.

Mesh algorithm generation

The software offers two types of mesh generation algorithms : the **Frontal isotropic** algorithm and the **Delaunay anisotropic** algorithm.

The **Frontal isotropic** algorithm uses only one spacing parameter and tries to generate triangles as regular as possible.

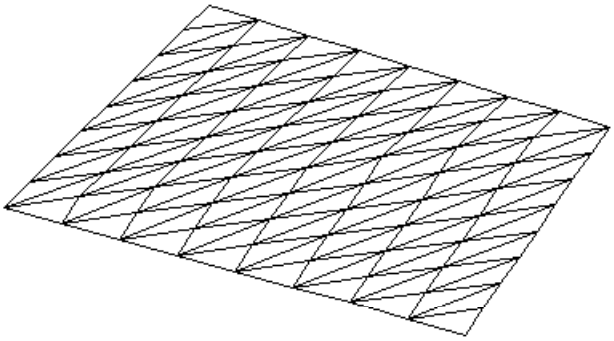
Figure 7 Mesh generated by the Frontal isotropic algorithm



This mesh was specified with a spacing of 40 meters on all the vertexes.

The **Delaunay anisotropic** algorithm uses three parameters (spacing for the major axis and for the minor axis, and a tilt angle for the major axis). This algorithm generates elements which are orientated in the designated direction with a mean spacing different for height and width. This algorithm is better adapted for river modeling since it maximizes the number of elements perpendicular to the flow without excessively increasing the total number of elements.

Figure 8 Mesh generated by the Delaunay anisotropic algorithm



This mesh was specified with a spacing of 200 meters at the major axis, a spacing of 40 meters at the minor axis, and a tilt angle of 15 degrees.

Select the type of algorithm in the group **Type of mesh** in the parameters window.

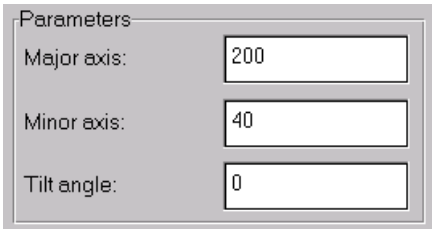
Default parameters

The default parameters for the mesh partition specify the default values to be applied on all the new surfaces that will be created. In fact, these parameters are applied on each vertex of a surface.

The default parameters also depend on the type of generation algorithm selected. If the **Frontal isotropic** algorithm is selected, only the mean spacing on the major axis is applicable while with the **Delaunay anisotropic** algorithm, the spacing for the major axis and for the minor axis, as well as the tilt angle are applied. (See the section *Mesh algorithm generation*).

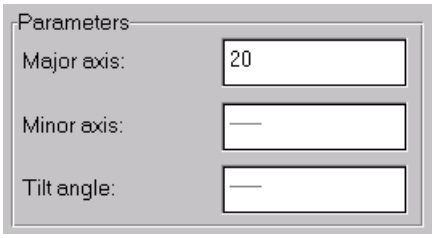
Click on the function button in the mesh partition control tool bar. The mesh parameters pop-up window will appear. To access the default parameters specification, you must not select any element of the partition. The title of the window should be **Default parameters**.

Figure 9 Parameters page for the *Delaunay anisotropic*



In the case of the Delaunay anisotropic, the parameters page allows entries in three fields.

Figure 10 Parameters page for the *Frontal isotropic*



In the case of the Frontal isotropic, the parameters page allows only an entry in the first field.

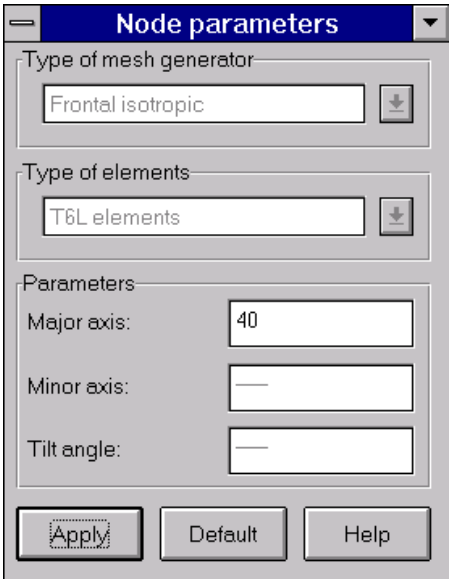
Notes :

- When you select a different type of mesh generator, the fields are activated/deactivated depending on the selection.
- When you click on the command button **Apply** to confirm the entries, there is always a validation of the content of the fields. You will be informed of any anomaly.

Parameters specific to a vertex

The specific parameters are carried by each vertex of the mesh partition. Since the type of mesh generator and the type of element are global characteristics, they cannot be modified and these fields are deactivated when you enter the node parameters mode.

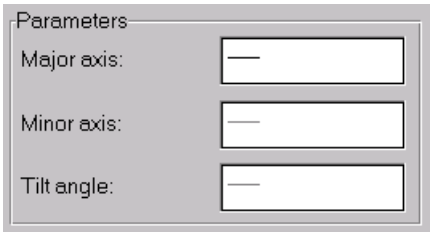
Click on the function button in the mesh partition control tool bar. The mesh parameters pop-up window will appear. To access the specification of parameters specific to a vertex, select an element of the partition (vertex, edge, or surface). The title of the window should be **Node parameters**.



Notes :

- You can select many vertex at a time, in which case you modify the specific parameters and apply the modification by clicking on **Apply**. The modification is applied immediately. You select a vertex by clicking on it in the display window and the parameters window is updated.

- You can also select an edge or a surface to apply a value on several vertexes at a time. To select an edge, click directly on the line ; to select a surface, click in the center of the surface. If the vertexes in the selection do not share the same values, the fields are filled by dashes.



  However, you can replace the dashes by a value which will be assigned to all the vertexes of the selection.

- You can also click on the command button **Default** to fill the fields with the default values of the partition. Click on the command button **Apply** to confirm the entry.

Generate a mesh on a surface

The generation of a mesh on a surface only allows you to validate your specifications. You can easily see if the mesh generated will satisfy your needs for this surface.

To generate a mesh on a surface :

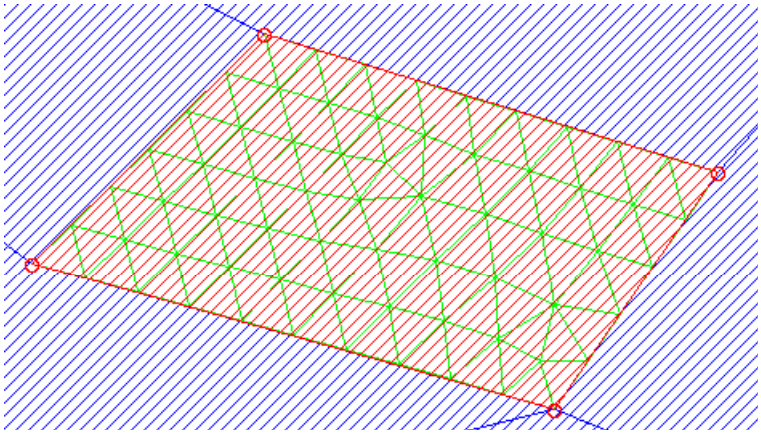
1 - Select a surface by clicking in its center in the display window.

2 - Click on the function button in the mesh partition control tool bar.

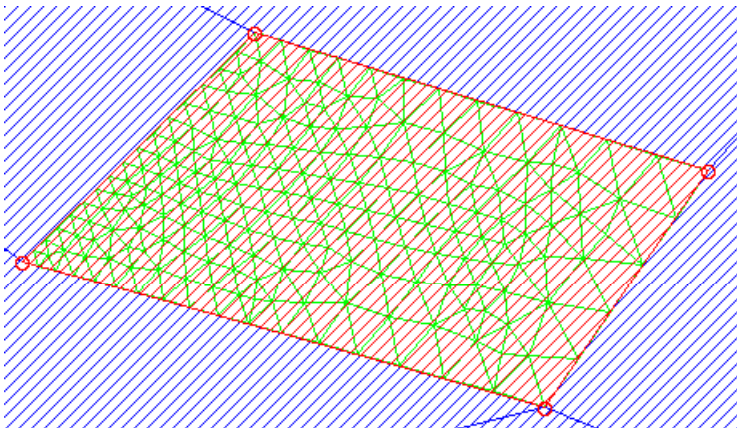
3 - An hourglass appears, and when it disappears, the result is displayed directly on the surface selected.

4 - To generate meshes on several surfaces at the same time, maintain the **Ctrl** key of the keyboard down during step 1.

Take for example the following partition where the initial surface has a spacing of 50 meters on all the vertexes (Frontal isotropic mesh generator).



With the generation tool, the results are seen immediately. It is then decided to refine the mesh on the left of this surface, Therefore we apply a spacing of 15 meters on the vertex at the extreme right and a spacing of 25 meters on the vertex at the top. We immediately generate the mesh on the surface.



We observe that the mesh is very refined at the locations that we have specified. You can repeat this operation at will, until you are fully satisfied with the results.

You can now proceed with the assembly of the mesh. (See the section *Assemble the global mesh*.)

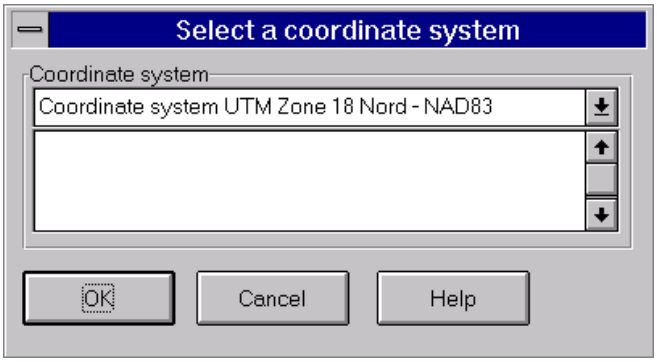
Assemble the global mesh

Now that you have correctly specified the parameters for your mesh partition and that you have validated the mesh generation on surfaces, you can proceed with the assembly of a new mesh.

To assemble a mesh from your mesh partition :

1 - Click on the function button in the mesh partition control tool bar.

2 - You will then be asked to choose a coordinate system to which the new mesh will be linked. The **Selection of a coordinate system** dialog box will appear.



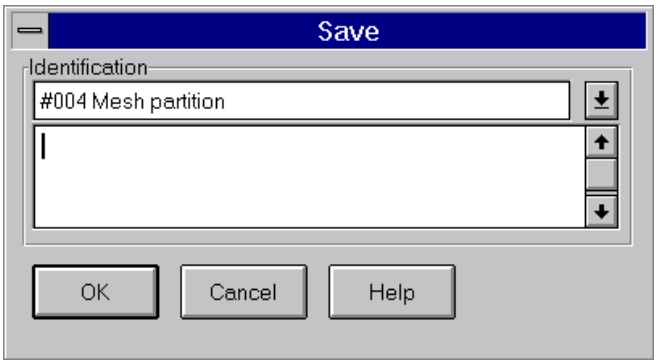
In the drop-down list, select a coordinate system. The list contains all the coordinate systems available in your project. **Warning :** if you want to launch simulations with the new mesh, you must absolutely select a **cartographic** coordinate system because the simulator must have spatial data

specified in meters (and not in degrees as in the case of geographic coordinate systems).

Click on **OK** to confirm the selection or on **Cancel** to cancel the mesh assembly.

3 - An hourglass appears and the mesh assembly begins. This operation can be rather long if the mesh contains a large number of nodes.

4 - When the operation is completed, a **Save** dialog box will appear. You are asked to save the new mesh.

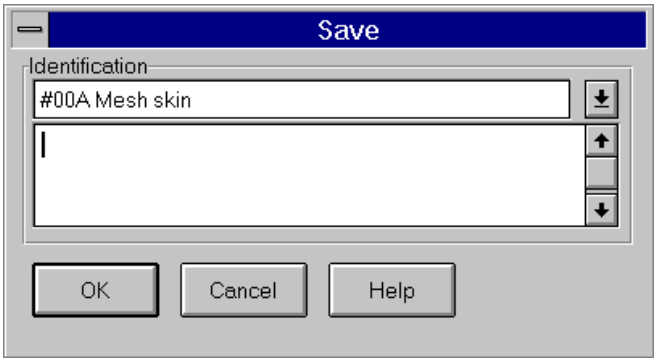


Enter the label and the description for the new mesh in their respective fields. Click on **OK** to save the mesh or on **Cancel** to cancel the save operation.

5 - After the mesh has been saved, the software will ask you if you want a mesh skin. The skin is a special entity representing the mesh skin elements, i.e. the contour outline and the islands, if any, of the domain. The **Creation of a mesh skin** dialog box will appear.

Click on **Yes** if you want a mesh skin and on **No** if you do not want one.

6 - If you have clicked on **Yes**, a **Save** dialog box appears to save the mesh skin.



Enter the label and the description for the mesh skin in their respective fields. Click on **OK** to save the mesh skin or on **Cancel** to cancel the save operation.

7 - You now have in your project an entity which can be displayed and used for the rest of your modeling work.

Warning :

- Only the meshes known as partition meshes can be used to conduct hydrodynamics simulations, not because the other meshes are not valid, but because the specification of boundary conditions is validated from the mesh partition from which the meshes were generated. Thus, no boundary conditions, no simulation. (See *Specify the boundary conditions*.)

Projection of the topography

The second very important step for the design of a numerical terrain model is the projection of the topography on the simulation mesh.

In real hydrodynamics studies, you will realize that you have to handle data from various sources, especially for topography.

Let us take for example the St. Lawrence River in Canada, rightly considered as a major river. The river must satisfy several different uses, such as commercial shipping, yachting, natural habitats, water intakes, wastewater disposal, etc.

The river has been, and still is, studied and surveyed. The sources of topography data are numerous and of variable quality. The commercial shipping channel is well documented, with dense and precise numerical data from the Canadian Coast Guard. The Canadian Hydrographic Service (CHS) also provides data outside the shipping channel. The Quebec Department of the Environment has data on the flood plains. Some riverside municipalities may have specific data for shore management, marinas, water intakes and wastewater disposal. Data is often lacking on the topography of the flood plains and the information must be gathered from maps or through field surveys.

To obtain a global overview of the topography, the data from these various sources must be integrated and projected on a common medium, the simulation mesh.

The topography projection mode provides the tool to integrate the data from various sources, using the partition as specification tool. For more information on the partition, see *Specification tool : the partition*.)

The software will allow you to simulate modifications to the topography to assess the impacts of various infrastructures and shoreline works. You can simulate digging a channel or installing booms, simply by modifying the local topography.

The following steps are necessary to process topography data :

1 - Import the sources of topography data

2 - Build the topography partition

3 - Specify the topography parameters

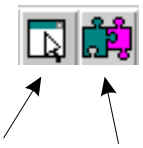
4 - Assemble the global topography

Note :

- The management of topography partitions and the edition of the final partition are treated in the section *Specification tool : the partition*.

In addition to the topography partition management and edition tool bar, there are two additional buttons to control the topography partition.

Figure 11 Partition control tool bar



Display partition parameters pop-up window

Assemble the result on the selected mesh simulation

Import the sources of topography data

Topography data from various sources must be imported in the database for efficient use.

Data importation and exportation procedures are explained in detail in the documentation on **Project management.** This section briefly summarizes topography data and entities.

External topography data, which are likely to be abundant, must be structured as follows. You need **data points** representing all the measurements available and a **scalar field - topography terrain** containing the topography for each measurement point.

The **data points** can be specified in the coordinate system of your choice (UTM, MTM, longitude/latitude, non georeferenced cartesian...). You import the **data points** in the database by linking with the coordinate system of your data.

Note :

- When importing the **data points**, the software will ask you if you want a **skin** on the data points. It is strongly recommended to generate a skin. It will make your work easier when building your topography partition, because you will have to locate the zones of your sources of topography data. The skin is more easily displayed than a cloud of points.

Next, you import the **scalar field - topography terrain** by linking to the **data points** that you have just imported. The software will automatically generate another entity, the **topography terrain**. You will be asked to provide a label and a description for this entity when you import the **scalar field - topography terrain**.

Note :

- For more details, read the documentation on **data format** and on the **data model** of the software.

The **topography terrain** is the abstraction of a source of topography data in your project ; it is with this type of entity that you will interact in the topography partition. (See the section *Specify the topography parameters*.)

Build the topography partition

Before building the topography partition, you must have imported all your sources of topography data. (See the section *Import the sources of topography data*.)

The documentation on partition management and edition is presented in the section *Specification tool : the partition*. This section describes the conception strategy for the topography partition.

You most likely have already generated one or several simulation meshes in the mesh generation mode. (See *Generate a mesh*.) The objective is to project the topography data on one of these meshes ; you must have topography data covering the **entire domain** of this mesh.

Before building your partition, you should display, using the static display tool (See the documentation on *data display*), the skin of your simulation mesh to set the entire domain. You should also display the skin of your data points corresponding to each of your sources of topography data.

You are now ready to build your topography partition. The objective is to build surfaces corresponding to your various sources of data.

Restriction :

- Your sources of data must always overlap slightly to successfully join these topography data.

For each surface of the partition, you will have to specify from which **topography terrain** you want to extract the data (See the section *Specify the topography parameters*.)

Note :

- You must absolutely cover the entire domain of the simulation mesh to be able to project the topography.

Specify the topography parameters

After you have graphically determined the different surfaces of the topography partition, you must specify the **topography terrain** to assign to each surface. (See the section *Build the topography partition*.)

The pop-up window to enter the topography parameters is accessible by two different means :

1 - By double-clicking on the target surface.

2 - By activating the function button in the partition control toolbar, at the bottom of the screen.

The pop-up window appears on the screen. It has two modes :

- Default parameters
- Surface parameters

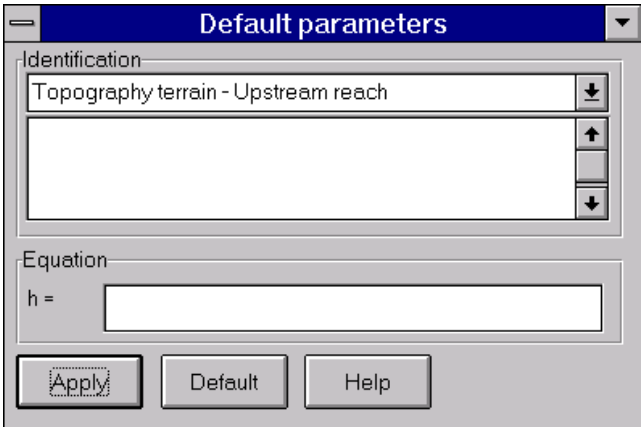
Note :

- With this window, you will have to specify the **topography terrain** to associate with each surface.

Default parameters

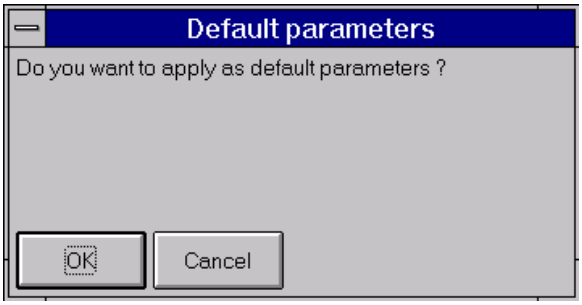
To specify the default parameters of the topography partition, you must not have selected any surface on the partition, i.e. click beside the topography partition in the display window.

The title of the window should be **Default parameters**.



You can use the default parameters to automatically assign these parameters to any new surface subsequently created.

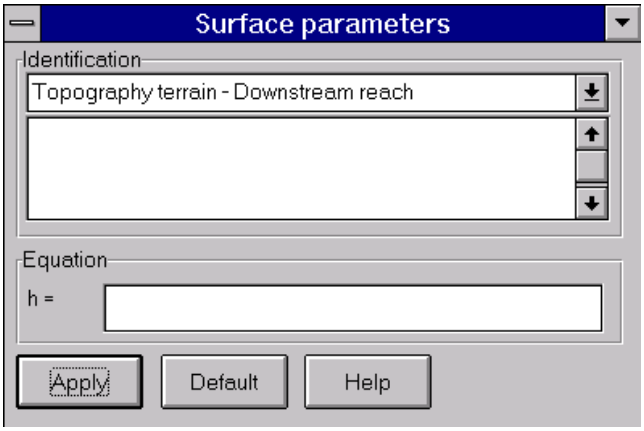
For details on these parameters, see the section *Surface parameters*. After you have specified the parameters, click on the command button **Apply** to confirm your entry. A confirmation dialog box will appear.



Click on **OK** to apply as default parameters or on **Cancel** if you do not want the default parameters.

Surface parameters

To specify the parameters specific to a surface in the topography partition, you must display the parameters pop-up window by clicking on the function button in the partition control tool bar. Next, select a surface using the mouse. The title of the pop-up window should be **Surface parameters**.



You have the possibility to specify the topography parameters for this surface. You can select the **topography terrain** to be associated with the surface and/or specify an equation defining the topography.

The possibilities are :

1 - Specify a topography terrain only

2 - Specify an equation only

3 - Specify a topography terrain and an equation modifying this topography.

Click on the command button **Apply** to apply your specification. You can also click on the command button **Default** to obtain the default parameters and apply them by clicking on **Apply**.

Specify a topography terrain only

You can specify a **topography terrain** only for the surface selected. Follow these steps :

1 - In the drop-down list, select the desired **topography terrain**. The list contains all the topography terrain imported in your project. (See the section *Import the sources of topography data*.)

2 - Click on the command button **Apply** to confirm your entry.

Specify an equation only

You can specify an equation only for the surface selected. You will then obtain an analytical topographic plan, such as a fixed elevation on the surface. Follow these steps :

1 - In the topography terrain drop-down list, select the option NIL which means no **topography terrain**.

2 - In the text field equation, enter the equation to be applied on the surface. For example, enter the value 92.45 meters. A part of the equation is implicit since there is a **z =** before the field. The complete equation will become **z = 92.45**.

3 - Click on the command button **Apply** to confirm your entry.

Specify a topography terrain and an equation modifying this topography

You can specify a **topography terrain** and an equation modifying this topography. Thus, you could dig a channel on the existing topography to assess the impacts on the flow. Inversely, you could also simulate an increased elevation of the topography to assess the installation of a boom. Follow these steps :

1 - In the drop-down list, select the desired **topography terrain**. The list contains all the topography terrains imported in your project. (See the section *Import the sources of topography data*.)

2 - In the text field equation, enter the equation to be applied on the surface. For example, you could remove 2 meters from the topography on this zone : **z - 2**. A part of the equation is implicit since there is a **z =** before the field. The complete equation will become **z = z - 2**, digging a channel 2 meters deep in the existing topography.

3 - Click on the command button **Apply** to confirm your entry.

Note :

- More complex manipulations are possible. You will then have to write a function for the calculator.

Assemble the global topography

Once you have completely specified the domain covered by your topography partition, you can proceed with the assembly of a topography to be projected on the designated simulation mesh.

For this task, follow these steps :

1 - Click on the function button in the topography partition control tool bar.

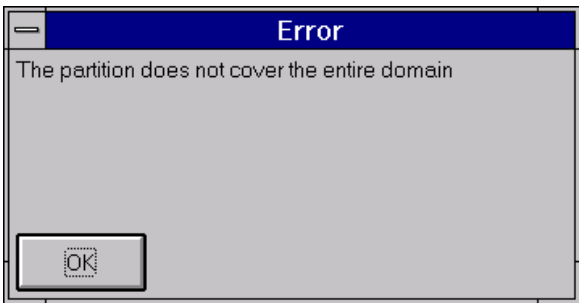
2 - The Select a mesh box appears.

In the drop-down list, select the simulation mesh desired. The list contains all the meshes and all the data points in

your project. Click on **OK** to confirm your selection or on **Cancel** to stop the topography assembly.

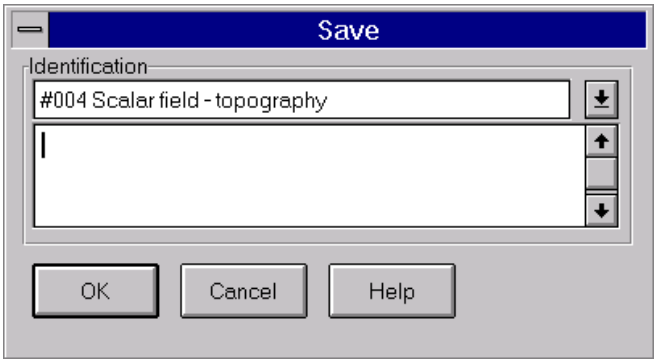
3 - An hourglass appears and the assembly begins. This task may be fairly long when there are several surfaces and several data sources. The density of the data points also affects the operation.

4 - If your topography partition did not cover the entire domain of the mesh selected, a warning box with an error message will appear.



5 - Click on OK to close the box. The software will let you save nevertheless at step 6.

6 - A dialog box will ask you to save a new entity of the type **scalar field - topography**. This entity is the topography projected on the mesh selected.



Enter a label and a description in their respective fields. Click on **OK** to save the entity or on **Cancel** if you do not want to save it.

7 - You now have an entity **scalar field - topography** which can be used for simulations with the associated mesh.

Hint :

- To reduce the time of assembly for the topography, you can use the data probe to view each **scalar field - topography terrain**, i.e. the source data specified for the surfaces in the topography partition. Adding each of these fields in the probe forces the software to load the related mesh and to build a localization table on it. The table is used to project data from a mesh to another. These meshes and localization tables will remain in the system memory, thus reducing considerably the assembly time for the topography.

Notes :

- At this stage, you should make sure that the topography projection gives a cohesive result, not far from the original data.
- To validate the topography, you can display your sources of data and your projected topography as isosurfaces with the same color scale. By visually comparing the results, you can at least detect gross anomalies..
- There is always an error induced during the projection. It depends on the density of the mesh and the density of your data sources. The user should remain aware of the error induced in the model.

Specify the boundary conditions

To be able to launch hydrodynamics simulations, boundary conditions must be specified. The boundary conditions apply here in the context of a **stationary** hydrodynamics simulation. For the present, the **non stationary** case is not addressed in the software.

When specifying boundary conditions, the idea is to impose a particular hydraulic event at the domain boundaries, i.e. the flow entering the model and the flow leaving the model.

To create a boundary conditions partition, you must therefore link yourself to a specific domain. Thus, you are asked to select a mesh partition. The mesh partition defines the domain and all the meshes generated from this partition will benefit of what will be specified in the boundary conditions partition.

To create a boundary conditions partition, follow these steps :

1 - In the **Modeler** menu, select the item **Boundary conditions [...]**.

2 - You then enter in the boundary conditions partition mode. Click on the function button in the partition management tool bar.

3 - A Selection of a mesh partition dialog box will appear.

In the drop-down list, select the mesh partition desired. The list contains all the mesh partitions of your project. Click on **OK** to confirm your selection or on **Cancel** to cancel the creation of a boundary conditions partition.

4 - The boundary condition partition is automatically generated from the **skin** of the mesh partition. The skin is the contour outline of the partition, as well as the outline of any island within.

5 - You have now to specify the boundary conditions parameters at the wanted locations on the edges of this skin. See the section *Specify the boundary conditions parameters*.

6 - Finally, you must generate boundary conditions by selecting a specific mesh. See the section *Specify the boundary conditions parameters*.

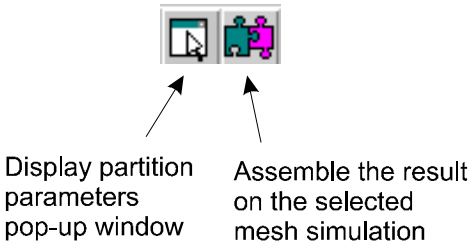
7 - You no have a new entity of the type **boundary conditions** which allows to launch hydrodynamics simulations.

Notes :

- There are no edit tool bar in the boundary conditions partition because the geometry is automatically generated as a function of the mesh partition selected..
- The partition management functions are described in the generic section *Specification tool : the partition*.

You have access to a boundary conditions partition control tool bar.

Figure 12 Partition control tool bar



You can use the following functions :

- Specify the boundary conditions parameters
- Generate the boundary conditions.

Specify the boundary conditions parameters

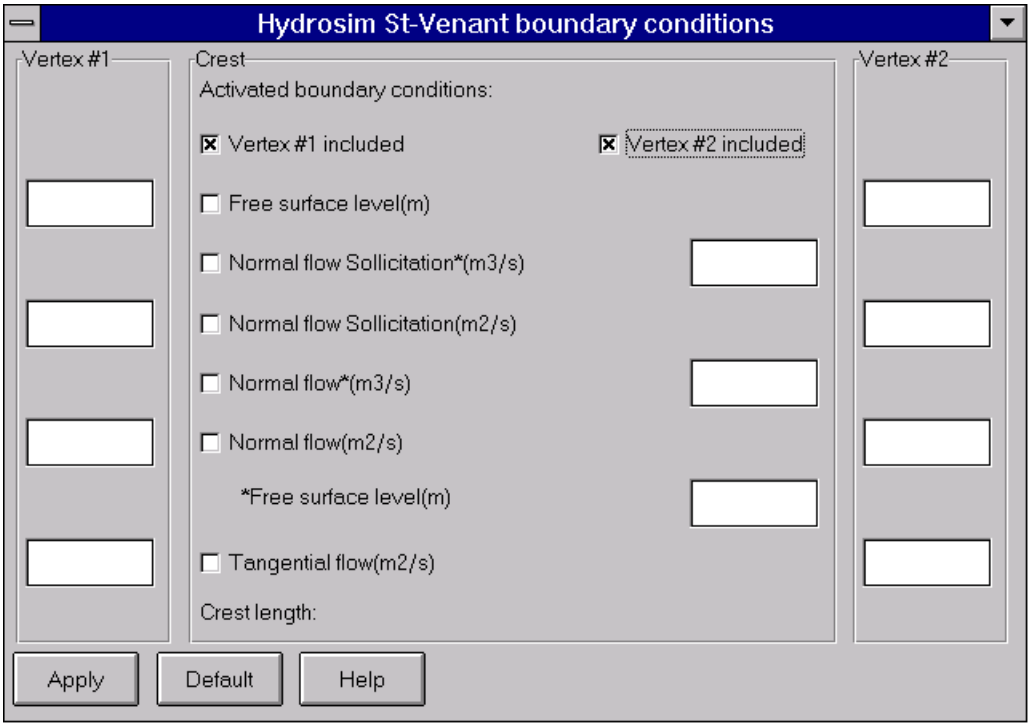
The specification of the boundary conditions parameters always applies on an edge of the partition.

Note :

- By default, nothing is specified, which means that the flow is tangent to the edge ; therefore, water neither enters nor exits. You must concentrate only on edges where water enters and where water exits.

You can specify the boundary conditions for HYDROSIM St-Venant in the boundary conditions parameters pop-up window.

To display the parameters pop-up window, click on the function button in the boundary conditions partition control tool bar. The pop-up window appears.



The boundary conditions parameters pop-up window contains three large zones :

1 - A zone of fields for vertex #1

2 - A zone of fields for vertex #2

3 - A zone of selection and specification of criteria.

On a vertex, you can specify the following constraints :

- Level of the free surface (m)
- Solicitation in normal discharge (m^2/s)
- Normal discharge (m^2/s)
- Tangential discharge (m^2/s)

These constraints, as applied on a vertex, may contradict each other ; but they only apply when they are activated on an edge. The constraints of a vertex are in fact shared by the two adjoining

edges. A constraint is always activated on an edge ; therefore you can specify everything that is necessary on the vertex to be useful for the two adjoining edges.

On an edge, the user can activate the desired constraints as specified on a vertex. The edge naturally validates the coherence between the constraints activated. Here are the valid combinations on an edge :

- Level of free surface
- Solicitation in normal discharge
- Normal discharge
- Tangential discharge
- Level of free surface AND normal discharge
- Level of free surface AND tangential discharge
- Solicitation in normal discharge AND tangential discharge
- Normal discharge AND tangential discharge

You can also specify the solicitation in normal discharge and the normal discharge in cubic meters per second (m^3/s). The length of the edge can be used to calculate a distribution of the discharge. The software integrates the constraint on the entire edge and distributes the result on the mesh nodes present below the edge.

It is possible to either include or exclude a vertex in the domain covered by the edge ; this allows you to deal with cases where vertexes may be included by two edges, one with an open boundary and the other with a closed boundary.

To specify the parameters on an edge :

1 - Select the edge desired and the parameters window is updated according to the constraints specified on the edge.

2 - Enter the desired parameters according to the above-mentioned constraints.

3 - Click on the command button **Apply** to confirm the entry.

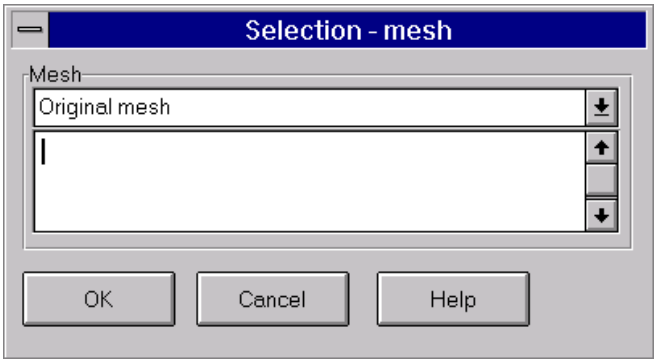
Generate the boundary conditions

Once you have completed the specification of boundary conditions parameters, you must generate the boundary conditions for the simulation mesh selected.

Follow these steps :

1 - Click on the function button in the boundary conditions partition control tool bar.

2 - A mesh selection dialog box appears.

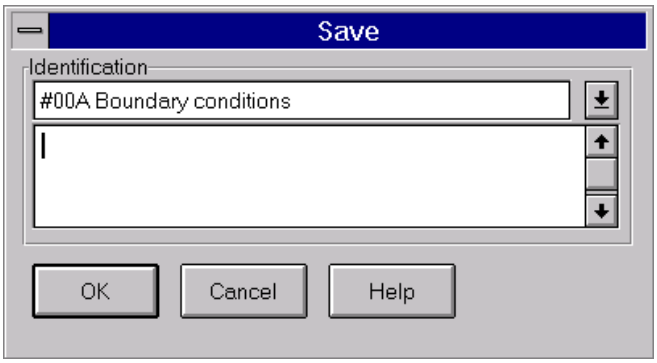


In the drop-down list, select the mesh of your choice. The list contains all the meshes which have been generated from the mesh partition selected during the creation of the boundary conditions partition.

Click on **OK** to confirm the selection or on **Cancel** to cancel the assembly.

3 - An hourglass appears and the generation of boundary conditions begins. It is possible that, at this stage, an error message is displayed to warn you of specification problems in the partition.

4 - When the assembly is completed, a save dialog box for the **boundary conditions** appears.



Enter the label and the description in their respective fields. Click on **OK** to confirm the save operation or on **Cancel** to cancel the operation.

5 - You now have a new entity of the type **boundary conditions** assembled on the mesh selected ; this entity can be used in the specification of a hydrodynamics simulation.

Specify the wind characteristics

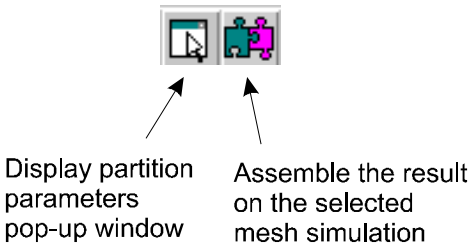
To specify the wind characteristics allows to modelize the constraints related to dominant winds when this factor must be taken into account. In fact, the wind can modify the natural discharge of a river under certain conditions.

To access the wind partition mode, select in the **Modeler** menu, the item **Wind [...]**.

You are now in the wind partition mode. Partition management and edition are described in the generic section *Specification tool : the partition*.

In addition to management and edition tool bars, there is a wind partition control tool bar.

Figure 13 Wind partition control tool bat



You have access to the following functions :

- Specify the wind parameters
- Generate a vector field - wind.

Specify the wind parameters

The wind parameters are always specified at the wind partition surfaces level. You can specify the parameters according to two modes :

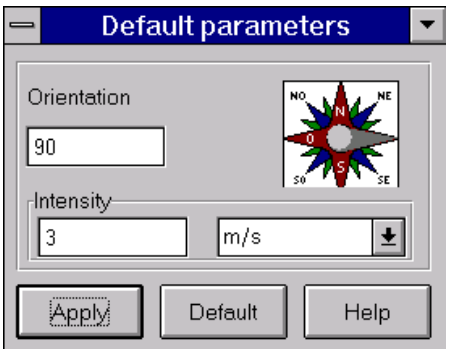
- Default parameters
- Surface parameters.

Default parameters

To specify the default parameters for the wind partition, follow these steps :

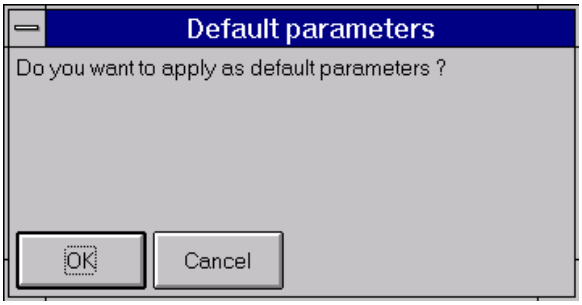
1 - Click on the function button in the wind partition control tool bar.

2 - A pop-up window appears. For the window to allow the specification of default parameters, you must not have selected a surface, i.e. click beside the partition in the display window. The title of the window should be **Default parameters**.



3 - The description of the fields appears in the section *Surface parameters*.

4 - Once you have specified the parameters, click on the command button **Apply** to apply these parameters as default parameters for all new surfaces created A confirmation box appears.



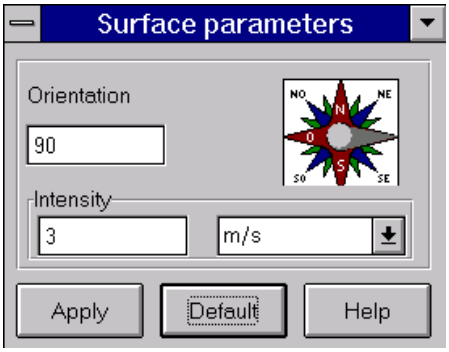
5 - Click on **OK** to apply the parameters as default or on **Cancel** to cancel the specification.

Surface parameters

To specify the parameters specific to a surface, follow these steps :

1 - Click on the function button in the wind partition control tool bar. A wind parameters pop—up window appears.

2 - To specify the parameters for a surface, click in the center of the chosen surface. The title of the window should be **Surface parameters**.



3 - You have to fill two main fields. The first field concerns wind **orientation**. You can specify it directly by a numerical value between 0 and 360. Another method consists in clicking on the gray cursor in the compass card and moving it to the appropriate direction. These fields are always synchronized to each other.

4 - The second field concern wind **intensity**. You can specify it by selecting in the drop-down list the measure unit to use. The list includes : knots, km/h and m/s. Next, enter an intensity value in the unit selected.

5 - Click on the command button **Apply** to apply these parameters to the surface selected.

6 - You can also click on the command button **Default** to obtain the default parameters, and then click on **Apply** to apply as surface parameters.

7 - Repeat steps 2-5 on all the surfaces.

Generate a vector field - wind

Once you have correctly specified the wind parameters on the surfaces of the wind partition, you can generate a vector field for wind.

Follow these steps :

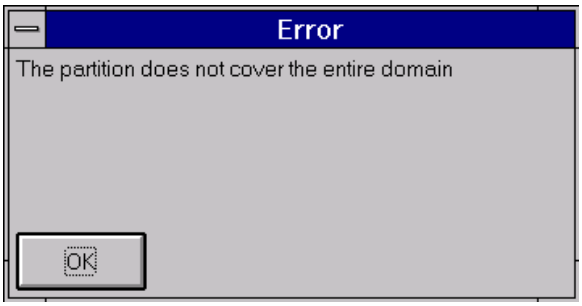
1 - Click on the function button in the wind partition control bar.

2 - A mesh selection box appears.

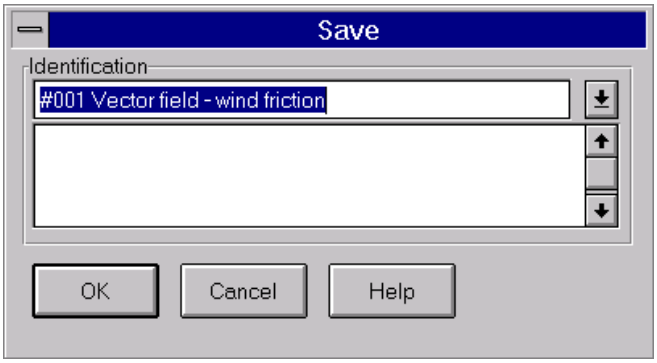
3 - In the drop-down list, select the simulation mesh desired. The list contains all the meshes present in your project. Click on **OK** to confirm the selection or on **Cancel** to cancel the generation of the **vector field - wind friction**.

4 - An hourglass appears and the generation begins. It is possible that you get an error message indicating that the partition does not cover the entire domain. This means that the domain of the mesh selected exceeds the domain of the

wind partition. Nevertheless, continue with step 5 and you can save the vector field for consultation.



5 - Once the generation of the **vector field - wind friction** is completed, a save dialog box for this field appears.



6 - Enter the label and the description for this new entity in the appropriate fields. Click on **OK** to confirm the save operation o on **Cancel** to cancel the operation.

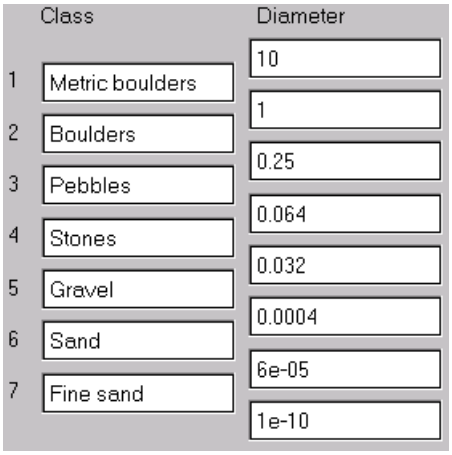
7 - You now have a new entity of the type **Vector field - wind friction** which can be used in a hydrodynamics simulation.

Specify the substrate characteristics

In the substrate mode, you can specify the soil composition of the river bed.

Substrate is characterized in the form of substrate categories divided in seven classes. Each class represents a grain size range (interval of diameters in meters). Here is an example of substrate classification :

Figure 14 Substrate classes



The substrate categorization contains 7 class labels and, for each class, a grain size range in meters.

An entity of the type **substrate classes** represents this classification in the database. The software does not allow the creation of this type of entity from a user interface. You must proceed through the importation of a classification. (See the documentation on **data formats**.)

MODELEUR provides two ways to specify the substrates on a domain. The first is to specify by surface the proportion of a substrate class observed over this zone ; the second is to use **substrate terrains** to specify by zone **data points** and values of **percentages of substrates** at these points. In the second case, you must proceed through the importation of **data points** and of **percentages of substrates** to generate **substrate terrains**.

To specify the substrates, you have to accomplish the following tasks :

1 - Determine the classification of the substrates

2 - Import substrate terrains (optional)

3 - Build a substrate partition

4 - Specify the substrate parameters

5 - Assemble the substrate on a designated mesh.

Note :

- This mode generates two types of entities : a **scalar field - substrate friction** for the hydrodynamics simulation and a **field - percentage of substrate** for the numerical terrain model.

Determine the classification of the substrates

Before modeling the substrate, you must determine which classification to use. It is important to classify correctly the substrates in function of what is being observed in the filed.

With the example of Rivière à la Truite, you have access to a classification used in actual real case studies. You can draw from it or use it as is. In this case, simply import this classification in your project. It is available in the terrain data directory in the sub-directory **terrain_rivière_truite**. The filename is **truite_classes_substrats.txt**.

Consult the documentation on **data format** for more information on the format required for the importation.

Import substrate terrains (optional)

The software allows to specify the substrates using **substrate terrains**. As with topography (see *Import the sources of topography data*), you can have **data points** to which are associated **fields - percentage of substrate**.

Details on data import and export operations are provided in the documentation on **Project management**. This section briefly summarizes the data and entities involved for the specification of the substrates.

External percentage of substrate data, which are likely to be abundant, must be structured as follows. You need **data points** representing all the measurements available and a **field - percentage of substrate** containing the proportions in each class (%) for each measurement points.

The **data points** can be specified in the coordinate system of your choice (UTM, MTM, longitude/latitude, non georeferenced cartesian...). You import the **data points** in the database by linking with the coordinate system of your data.

Note :

- When importing the **data points**, the software will ask you if you want a **skin** on the data points. It is strongly recommended to generate a skin. It will make your work easier when building your substrate partition, because you will have to locate the zones of your sources of topography data. The skin is more easily displayed than a cloud of points..

Next, you import the **field - percentage of substrate** by linking to the **data points** that you have just imported. The software will automatically generate another entity, the **substrate terrain**. You

will be asked to provide a label and a description for this entity when you import the **field - percentage of substrate.**

Note :

- For more details, read the documentation on **data format** and on the **data model** of the software.

The **substrate terrain** is the abstraction of a source of percentage of substrate data in your project ; it is with this type of entity that you will interact in the substrate partition. (See the section *Specify the substrate parameters*

Remarks :

- To this date, this method has not been used in real cases in the course of our studies. We do not pretend that it constitutes an interesting or desirable path. It is available for experimentation.
- There is an important limitation to the use of substrate terrains. You cannot validate, with the tools available, what has been modelized. You cannot trace isolines or isosurfaces representing percentages of substrate.

Build a substrate partition

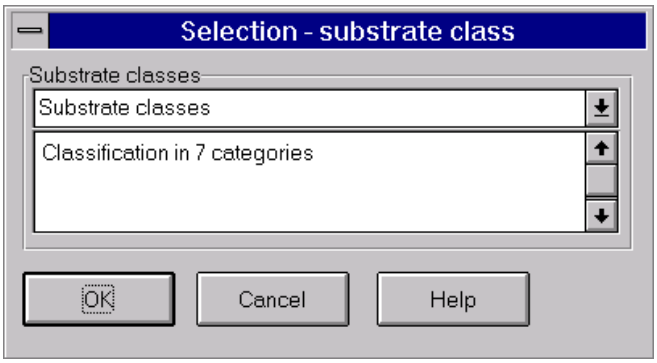
To build a substrate partition, select in the **Modeler** menu the item **Substrates [...]**. You are now in the substrate partition mode.

The documentation on partition management and edition is presented in the section *Specification tool : the partition*

To create a substrate partition, follow these steps :

1 - Click on the function button in the substrate partition management tool bar.

2 - A substrate class selection box appears.



3 - In the drop-down list, select the appropriate class. The list contains all the classifications available in your project. Click

on **OK** to confirm your selection or on **Cancel** if you do not want to create a substrate partition.

4 - The partition is then linked to the selected classification.

Specify the substrate parameters

In a substrate partition, the parameters are applied at the level of the surfaces of the partition. Two modes are available to specify the parameters :

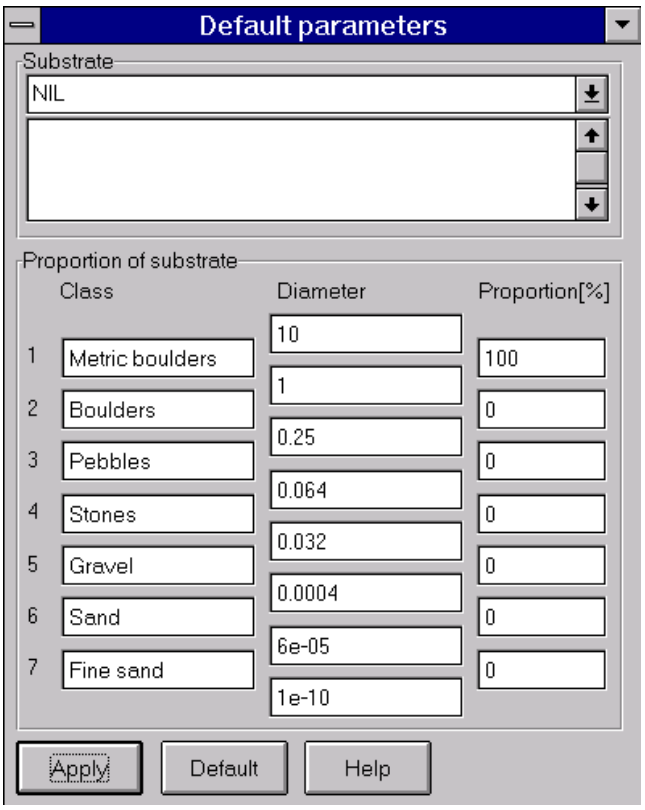
- Default parameters
- Surface parameters.

Default parameters

To specify the default parameters for a substrate partition, follow these steps :

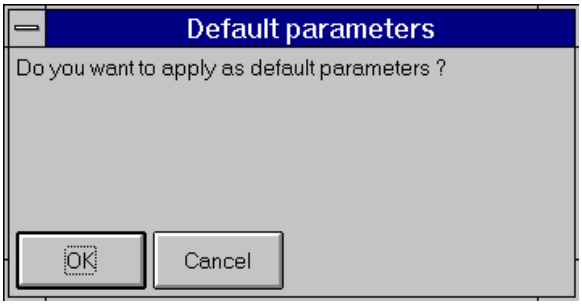
1 - Click on the function button in the substrate partition control tool bar.

2 - A pop-up window appears. To be able to specify default parameters in the window, you must not have selected a surface, i.e. click beside the partition in the display window. The title of the window should be **Default parameters**.



3 - The text fields are described in the section *Surface parameters*.

4 - Once the parameters have been specified, click on the command button **Apply** to apply these parameters as default parameters for all new surfaces created A confirmation box appears.



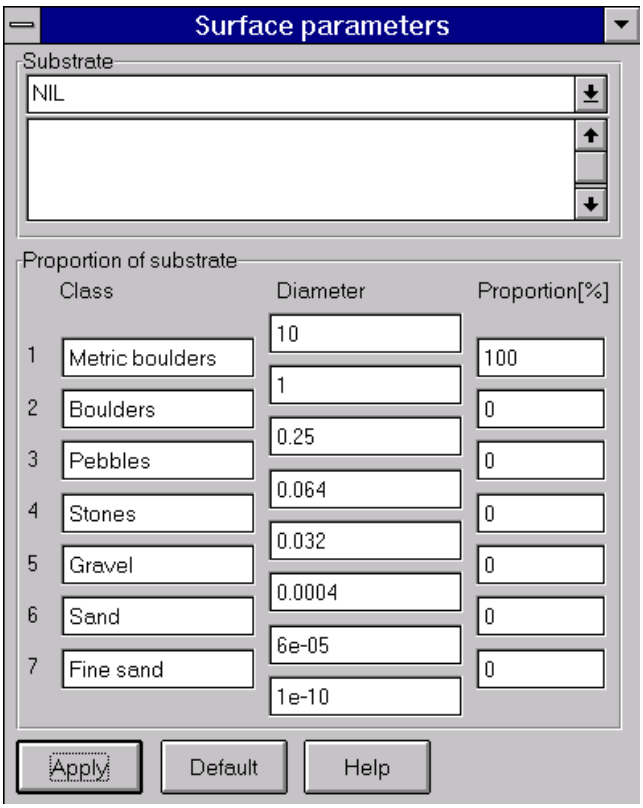
5 - Click on **OK** to apply the parameters as default or on **Cancel** if you do not want to apply them.

Surface parameters

To specify the parameters specific to a surface, follow these steps :

1 - Click on the function button in the substrate partition control tool bar. A substrate parameters pop-up window appears.

2 - To specify the parameters on a surface, click in the center of the surface chosen. The title of the window should be **Surface parameters**.

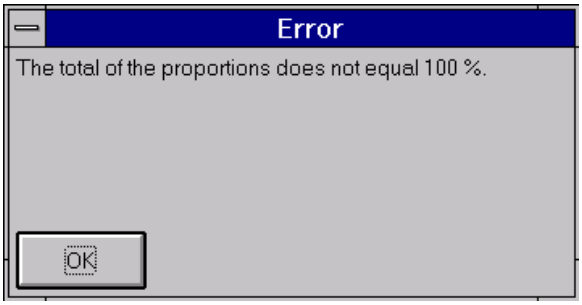


3 - You can function in two different modes. You can use the **substrate terrains** in the upper portion (see *Import substrate terrains (optional)* ; or you can specify directly the substrate proportions by surface. These two modes are mutually exclusive.

4 - If you want to specify a **substrate terrain**, select the substrate terrain of your choice in the drop-down list. The list contains all the substrate terrains present in your project.

5 - If you want to specify directly the proportions, select the NIL label (meaning no substrate terrain) in the drop-down list. Next, in the right lower section, specify the proportions, in percentage, observed on the surface. The total of the proportions must be 100 %.

6 - Click on the command button **Apply** to apply these parameters to the surface selected. At this stage, an error message will be displayed if the total of the proportions is not 100 %.



7 - You can also click on the command button **Default** to obtain the default parameters, and then click on **Apply** to apply as surface parameters.

8 - Repeat steps 2 to 6 for each surface.

Assemble the substrate on a designated mesh

Once you have correctly specify the substrate parameters on the surfaces of the substrate partition, you can generate the **scalar field - substrate friction** and the **field - percentage of substrate**.

Note :

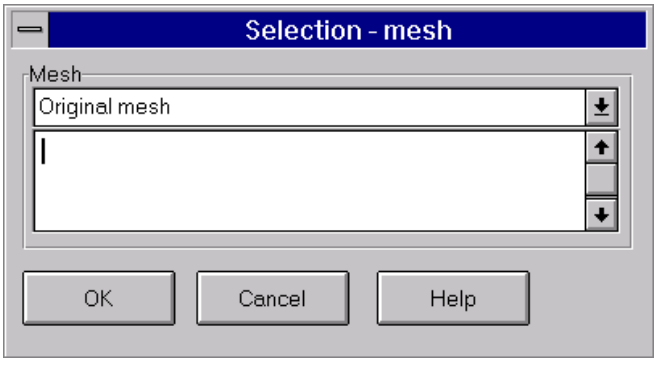
- For the **scalar field - substrate friction**, the software transforms the proportions of substrate according to categories in a friction coefficient usable by the hydrodynamics simulator. The formula used is :

```
if (diameterMedian > 0.7)
 friction = 0.051;
else
 friction =
1/((34.9(-log(diameterMedian)^0.31) + 0.0017)
```

To assemble the substrate, follow these steps :

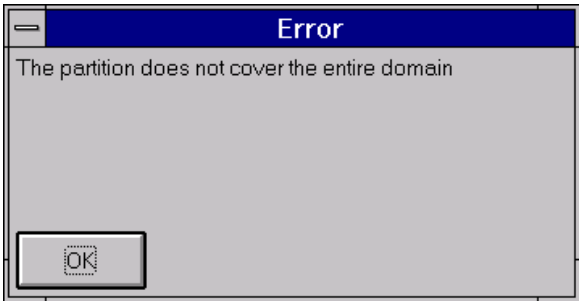
1 - Click on the function button in the substrate partition control tool bar.

2 - A mesh selection box appears.

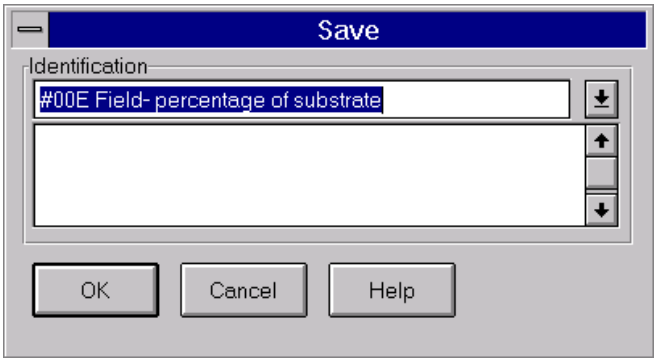


3 - In the drop-down list, select the simulation mesh of your choice. The list contains all the meshes available in your project. Click on the command button **OK** to confirm the selection or on **Cancel** to cancel the generation.

4 - An hourglass appears and the generation begins. It is possible that you get an error message indicating that the partition does not cover the entire domain. This means that the domain of the mesh selected exceeds the domain of the substrate partition. Continue nevertheless with step 5 and you will be able to save the fields for consultation.

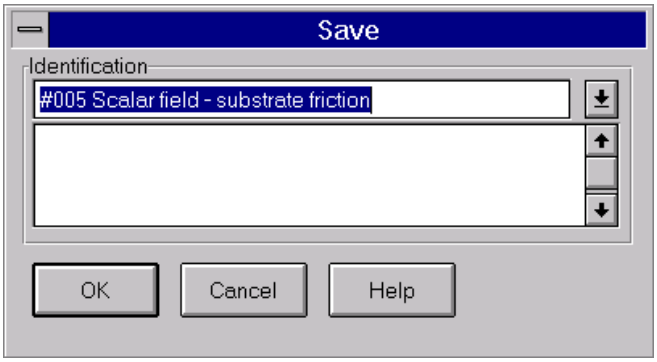


5 - Once the generation is completes, a **field - percentage of substrate** save dialog box appears.



6 - Enter the label and the description for this new entity in the appropriate fields. Click on **OK** to confirm the operation or on **Cancel** to cancel it.

7 - Next a **scalar field - substrate friction** save dialog box appears.



8 - Enter the label and the description for this new entity in the appropriate fields. Click on **OK** to confirm the operation or on **Cancel** to cancel it.

9 - You now have two new entities : one of the type **scalar field - substrate friction** which can be used in hydrodynamics simulations and one of the type **field - percentage of substrate** which will be added to the numerical terrain model.

Specify the ice characteristics

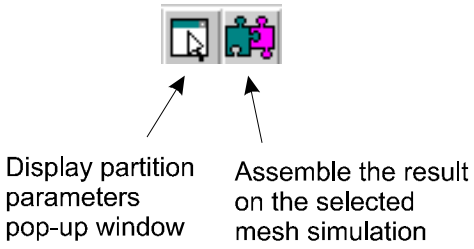
To modelize the constraints related to the presence of ice, you must specify the ice characteristics. Ice imposes two types of constraints : friction at the surface of the water and a flow restriction zone.

To access the ice partition mode, select in the **Modeler** menu the item **Ice [...]**.

You are now in the ice partition mode. The documentation on partition management and edition is presented in the section *Specification tool : the partition*.

In addition to management and edition tool bars, you have an ice partition control tool bar.

Figure 15 Ice partition control tool bar



You have access to the following functions :

- Specify the ice parameters
- Assemble a field for ice.

Specify the ice parameters

The ice parameters are always specified for surfaces of the ice partition. There two modes to specify the parameters :

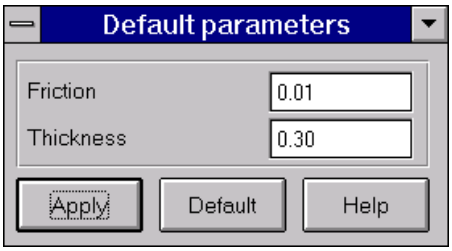
- Default parameters
- Surface parameters.

Default parameters

To specify the default parameters for the ice partition, follow these steps :

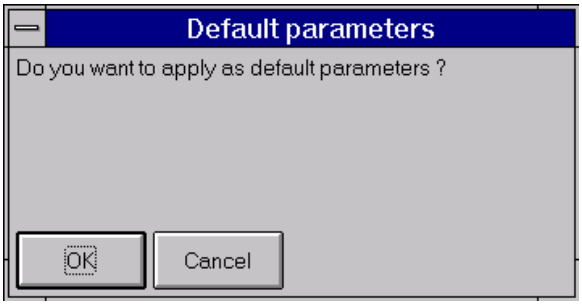
1 - Click on the function button in the ice partition control tool bar.

2 - A pop-up window appears. To be able to specify default parameters in the window, you must not have selected a surface, i.e. click beside the partition in the display window. The title of the window should be **Default parameters**.



3 - The text fields are described in the section *Surface parameters*.

4 - Once the parameters have been specified, click on the command button **Apply** to apply these parameters as default parameters for all new surfaces created A confirmation box appears.



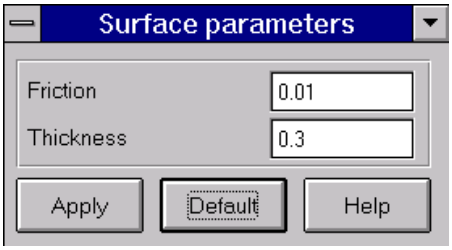
5 - Click on **OK** to apply the parameters as default or on **Cancel** to cancel the operation.

Surface parameters

To specify the parameters specific to a surface, follow these steps :

1 - Click on the function button in the ice partition control tool bar. An ice parameters pop-up window appears.

2 - To specify the parameters on a surface, click in the center of the surface of your choice. The title of the window should be **Surface parameters**.



3 - You have to fill two fields. The first field concerns the friction related to ice cover. You can assign a Manning friction coefficient to simulate the friction.

4 - The second field concerns the ice thickness on the surface to determine the flow restriction zone.

5 - Click on the command button **Apply** to apply these parameters to the surface selected.

6 - You can also click on the command button **Default** to obtain the default parameters and then click on **Apply** to apply as surface parameters.

7 - Repeat steps 2 to 5 on each surface.

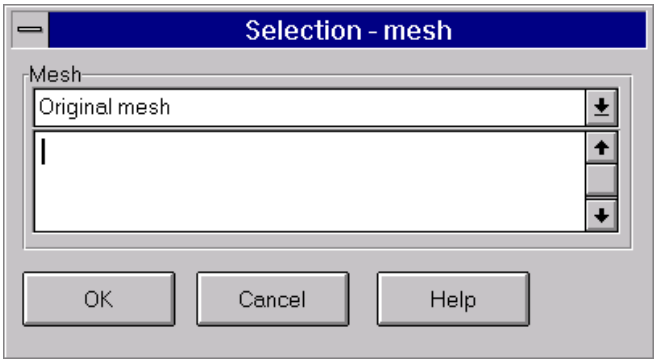
Assemble a field for ice

Once you have correctly specified the ice parameters on the surfaces of the ice partition, you can generate a **field - ice**.

Follow these steps :

1 - Click on the function button in the ice partition control tool bar.

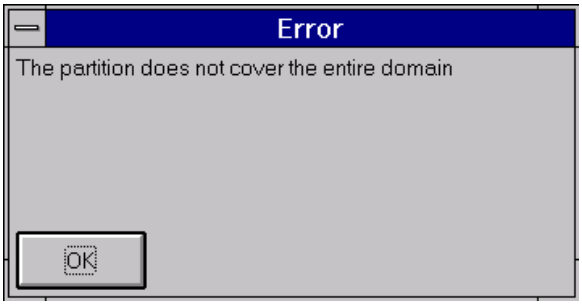
2 - A mesh selection box appears.



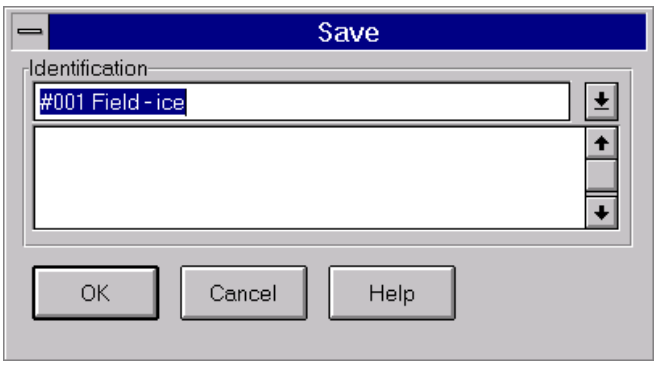
3 - In the drop-down list, select the simulation mesh of your choice. The list contains all the meshes available ion your project. Click on **OK** to confirm the selection or on **Cancel** to cancel the generation of the **field - ice**.

4 - An hourglass appears and the generation begins. It is possible that you get an error message indicating that the

partition does not cover the entire domain. This means that the domain of the mesh selected exceeds the domain of the ice partition. Nevertheless, continue with step 5 and you can save the field for consultation.



5 - Once the generation of the **field - ice** is completed, a save dialog box for this field appears.



6 - Enter the label and the description for this new entity in the appropriate fields. Click on **OK** to confirm the save operation or on **Cancel** to cancel the operation.

7 - You now have a new entity of the type **field - ice** which can be used in hydrodynamics simulations.

Specify the other friction coefficients

To complete the model, you must specify the other friction coefficients in the form of a Manning coefficient applied by surface.

Note :

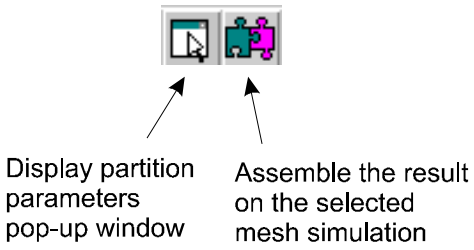
- It is recommended not to use this type of friction without a good reason. Try to specify your numerical terrain model with natural features such as substrate, wind or ice.

To access the friction partition mode, select in the **Modeler** menu the item **Friction [...]**.

You are now in the friction partition mode. The documentation on partition management and edition is presented in the section *Specification tool : the partition*.

In addition to management and edition tool bars, you have a friction partition control tool bar.

Figure 16 Friction partition control tool bar



You have access to the following functions :

- Specify the Manning friction parameters
- Assemble a scalar field - Manning friction.

Specify the Manning friction parameters

The Manning friction parameters are always specified on the surfaces of the friction partition. There are two modes to specify the parameters :

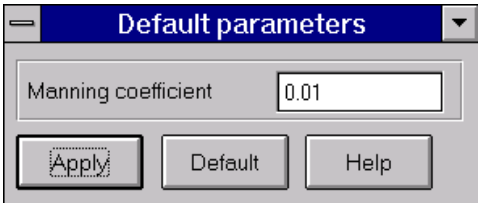
- Default parameters
- Surface parameters.

Default parameters

To specify the default parameters for the friction partition, follow these steps :

1 - Click on the function button in the friction partition control tool bar.

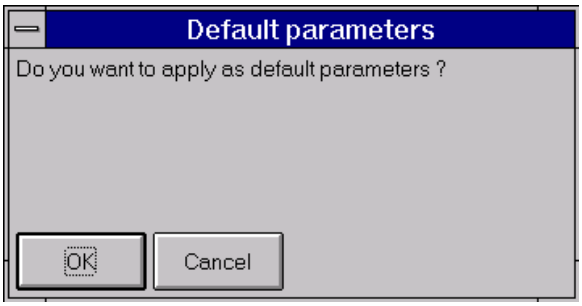
2 - A pop-up window appears. To be able to specify default parameters in the window, you must not have selected a surface, i.e. click beside the partition in the display window. The title of the window should be **Default parameters**..



3 - The text fields are described in the section *Surface parameters*.

4 - Once the parameters have been specified, click on the command button **Apply** to apply these parameters as default

parameters for all new surfaces created A confirmation box appears.



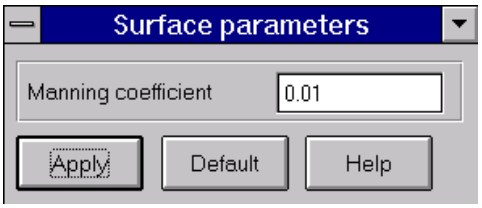
5 - Click on **OK** to apply the parameters as default or on **Cancel** to cancel the operation.

Surface parameters

To specify the parameters specific to a surface, follow these steps :

1 - Click on the function button in the friction partition control tool bar. A friction parameters pop-up window appears.

2 - To specify the parameters on a surface, click in the center of the surface of your choice. The title of the window should be **Surface parameters**.



3 - You must specify a friction coefficient applicable on the surface selected.

4 - Click on the command button **Apply** to apply these parameters to the surface selected.

5 - You can also click on the command button **Default** to obtain the default parameters and then click on **Apply** to apply as surface parameters.

6 - Repeat steps 2 to 4 on each surface.

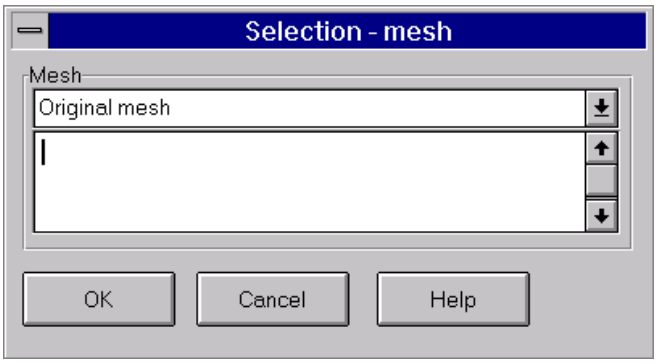
Assemble a scalar field - Manning friction

Once you have correctly specified the friction parameters on the surfaces of the friction partition, you can generate a **scalar field - Manning friction**.

Follow these steps :

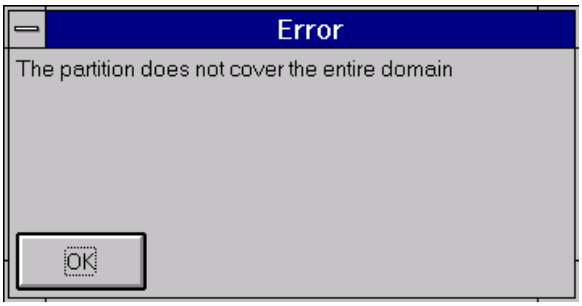
1 - Click on the function button in the friction partition control tool box.

2 - A mesh selection box appears.

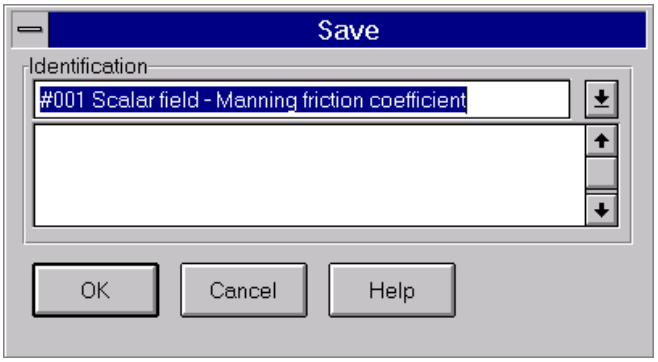


3 - In the drop-down list, select the simulation mesh of your choice. The list contains all the meshes available ion your project. Click on **OK** to confirm the selection or on **Cancel** to cancel the generation of the **scalar field - Manning friction**.

4 - An hourglass appears and the generation begins. It is possible that you get an error message indicating that the partition does not cover the entire domain. This means that the domain of the mesh selected exceeds the domain of the friction partition. Nevertheless, continue with step 5 and you can save the field for consultation.



5 - Once the generation of the **scalar field - Manning friction** is completed, a save dialog box for this field appears.



6 - Enter the label and the description for this new entity in the appropriate fields. Click on **OK** to confirm the save operation or on **Cancel** to cancel the operation.

7 - You now have a new entity of the type **scalar field - Manning friction** which can be used in hydrodynamics simulations.

Chapter 7 Conducting a simulation

With the software MODELEUR, you can specify, launch, recover and validate the results of a hydrodynamic simulation.

Prerequisite :

- You must have previously integrated the terrain data in your project. You must have generated meshes, transferred the topography on the meshes, specified the boundary conditions for various hydrological events, and specified the various friction coefficients.

You are now ready to execute the following tasks :

1 - Specify the characteristics of a simulation

2 - Launch and monitor a simulation

3 - Recover the results

4 - Validate the results.

Note :

- Loopbacks are frequent and certain steps may be done over and over until satisfactory results are obtained.

Specify the characteristics of a simulation

To specify the characteristics of a simulation, you must use the simulation pop-up window. To display this window, select in the **Simulation** menu the item **Stationary...**

The simulation pop-up window appears. There are three large zones in this window :

1 - Zone of the control buttons

2 - Zone of specifications display

3 - Simulation.

The following sections will help you understand how the interface works.

Zone of the control buttons

You have access to a function buttons tool bar to manage the simulation. The upper part of these buttons allows to :

- Create a new simulation
- Load a simulation
- Save the simulation
- Save the simulation under a different name (Save As...)
- Close the simulation

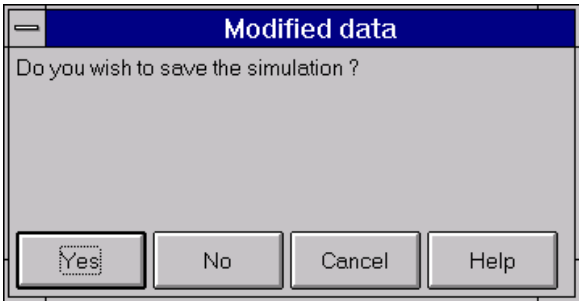
The other buttons will be dealt with in the section *Launch and monitor a simulation* and the section *Recover the results*.

Create a new simulation

To create a new simulation :

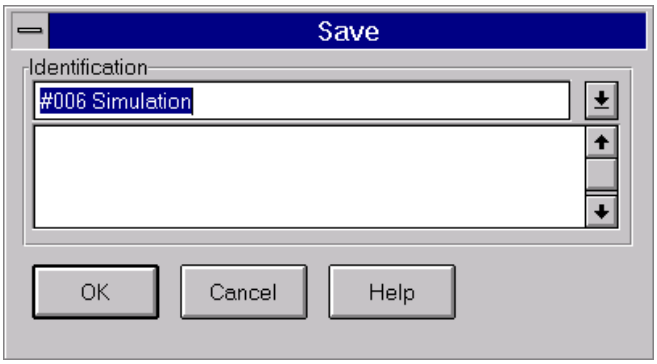
1 - Click on **New** in the button tool bar of the simulation window.

2 - If a simulation is already loaded and has been modified, the software will warn you about this situation.



3 - Click on **Yes** to save the simulation or on **No** if you do not want to save the modifications. Click on the command button **Cancel** if you do not want to create a new simulation.

4 - If this simulation is a new one, a simulation save box will be displayed.



5 - Enter the label and the description of your choice in the appropriate fields. Click on **OK** to confirm the save operation or on **Cancel** to cancel it and thus not create a new simulation.

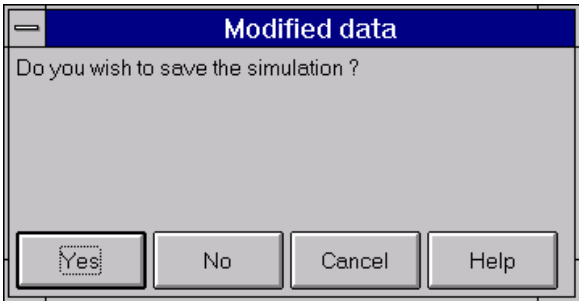
6 - A new simulation has now been created without any parameters having been specified. To specify the simulation parameters, see *Simulation*.

Load a simulation

To load an existing simulation :

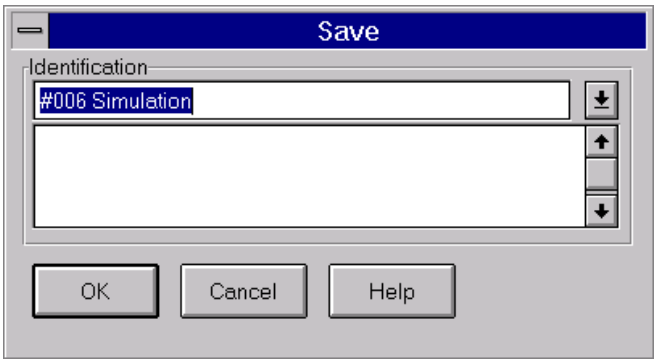
1 - Click on **Load** in the button tool bar of the simulation window.

2 - If a simulation is already loaded and has been modified, the software will warn you of the situation.



3 - Click on **Yes** to save the simulation or on **No** if you do not want to save the modifications. Click on the command button **Cancel** if you do not want to load the simulation.

4 - If this simulation is a new one, a simulation save box will be displayed.



5 - Enter the label and the description of your choice in the appropriate fields. Click on **OK** to confirm the save operation or on **Cancel** to cancel it and not load another simulation.

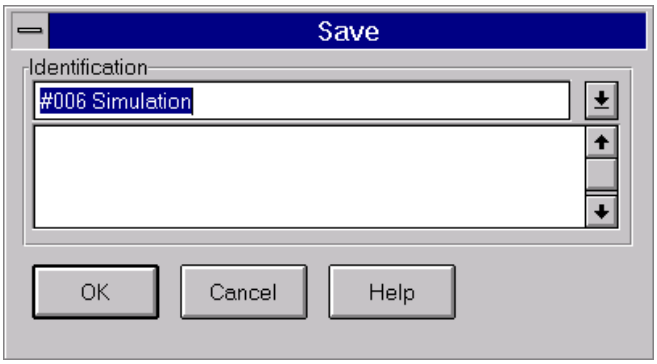
6 - The simulation is then loaded and the simulation window takes a configuration representing the content of the simulation.

Save the simulation

To save the current simulation :

1 - Click on the function button **Save** in the simulation window management tool bar.

2 - If the current simulation has never been saved, a simulation save box appears.



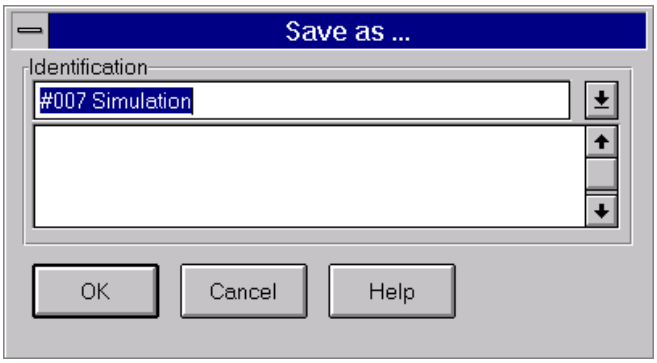
3 - Enter the label and the description of your choice in the appropriate fields. Click on **OK** to confirm the save operation or on **Cancel** to cancel it.

4 - The simulation is saved under its current state.

Save the simulation under a different name (Save As...)

To Save As... the current simulation :

1 - Click on the function button **Save as...** in the simulation window management tool bar. The simulation **Save As...** box appears.



2 - In the upper part of the section **Identification**, enter the name that you want to give to the copy of the simulation. The drop-down list shows you the names of the existing simulations so that you can draw from these. However the data model requires that the name of a simulation must be unique among all the simulations of the same type. A warning message will inform you if you do not respect this rule.

3 - In the lower part of the section **Identification**, you can enter a description of the simulation.

4 - Click on **OK**. A copy of the current simulation under its current state is created in the database and automatically becomes the current simulation.

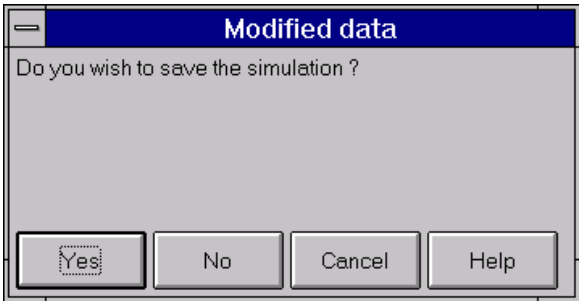
Note :

- If the simulation that you wish to save as... has never been saved before, a standard save operation will occur. (See *Save the simulation*.)

Close the simulation

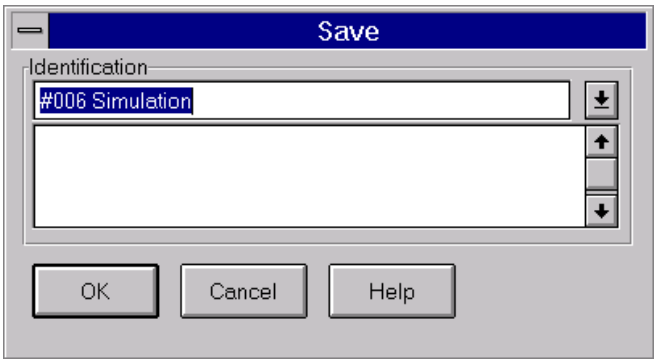
To close the simulation, you must close the simulation window using the button or the system menu.

1 - If the simulation has been modified, the software warns you of the situation.



2 - Click on the command button **Yes** to save the simulation or on **No** if you do not want to save the modifications. If you click on the command button **Cancel,** the closing operation is canceled.

3 - If this simulation is a new one, a simulation save box will be displayed.



4 - Enter the label and the description of your choice in the appropriate fields. Click on **OK** to confirm the save operation or on **Cancel** to cancel it and thus not close the simulation.

Zone of specifications display

The display zone in the simulation window allows to display the current composition of the simulation which is a list of commands specified in the parameters zone. (See *Simulation*.)

The display zone allows you to interact with the commands in various ways :

- Edit a command
- Remove a command
- Move a command.

Note :

- The simulation window may be enlarged using the mouse. Simply maintain the mouse clicked down on the outline of the window and drag i to obtain the size wanted. This offers a larger display area when needed.

Edit a command

To edit/modify an existing command :

1 - With the right mouse button, select the item to edit. A pop-up menu appears.

2 - Click on **Edit** in this menu.

3 - The corresponding page is displayed in the simulation parameters zone. You can modify or consult the parameters of this command.

4 - Click on the command button **Apply** in the simulation parameters page. The command will be updated.

Notes :

- A double-click on the command to edit will have the same result as the use of the pop-up menu.
- Consult the section corresponding to the command under edition to find the details. (See *Simulation*.)

Remove a command

To remove a command from the current simulation :

1 - With the right mouse button, select the command to remove. A pop-up menu appears.

2 - Click on **Remove** in this menu.

3 - The command will be removed from the list and from the actual simulation.

Move a command

The simulation commands are in an order important to the simulator. These parameters will produce commands which will be executed in the prescribed order.

To move a command, i.e. change its position in the list, follow these steps :

1 - With the right mouse button, select the command to be moved.

2 - Maintain the button press down and drag the command to the new position in the list. The cursor changes to a vertical arrow to indicate that you are in the displacement mode.

3 - Once you have reached the target position, release the mouse button. The command will be moved to the good position.

4 - Repeat the operation until you are in the right location.

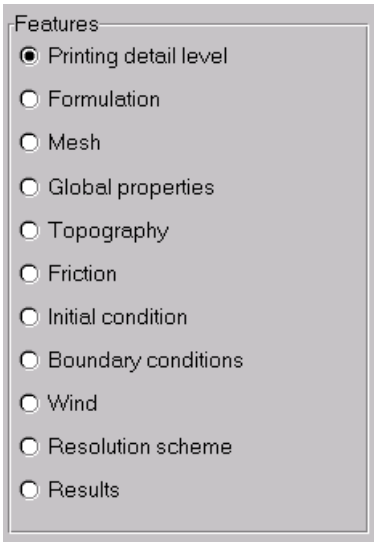
Note :

- The simulation window may be enlarged using the mouse. Simply maintain the mouse clicked down on the outline of the window and drag i to obtain the size wanted. This offers a larger display area when needed. This could be useful if you want to move a command to a position which is not visible.

Simulation parameters zone

The parameters zone of the simulation box is where you can specify the commands for the simulator..

In the left part of this zone, you have access to radio button to select the command to be specified. To display the page in the right part of the parameters zone, click on the button radio corresponding to the target parameter.



You must specify the following parameters to construct a coherent simulation for the simulator :

- Print detail level
- Formulation
- Mesh
- Global properties
- Topography
- Friction
- Initial conditions
- Boundary conditions
- Wind
- Resolution scheme
- Results.

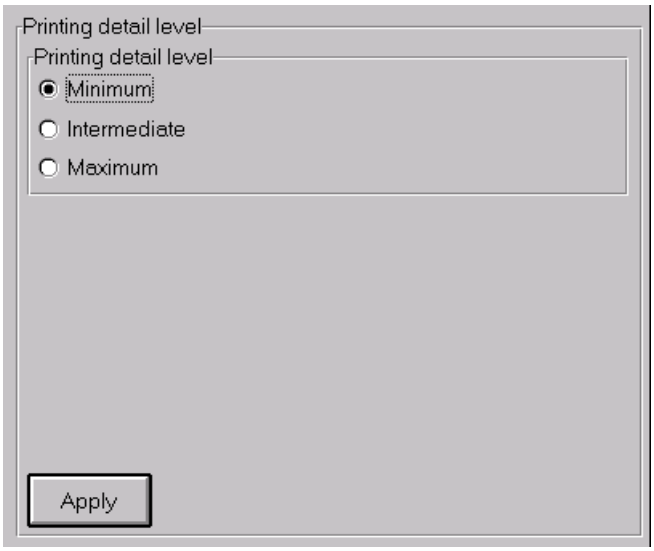
Note :

- After correctly specifying the parameters of your simulation, you are ready to launch the simulation. (See *Launch and monitor a simulation*.)

Print detail level

The print level parameter controls the detail level that the simulator will provide during the processing of the following commands. This command can be placed at several locations during the specification of the simulation commands to print a detail level specific to certain commands.

There are three levels of print detail : minimum, intermediate or maximum.



To add a command of print detail level :

1 - Select the level of your choice with the radio buttons of the print level parameters page.

2 - Click on the command button **Apply** to add the command at the end of the list of commands in the display list.

Note :

- You can modify, remove or move this command later, see *Zone of specifications* display.

Examples of uses :

Take for example the case where you want to make sure that the simulator reads correctly the coordinates and the elements of your mesh. You may have previously fixed the print level at minimum and you want to bring it back to minimum after reading of the mesh :

1 - Select the ***Print detail level*** page. Fix the level at **minimum**. Click on the command button **Apply**.

2 - Select the ***Formulation*** page. Select, for example, *T6L Drying/Wetting* in the drop-down list. Click on the command button **Apply**.

3 - Select the ***Print detail level*** page. Fix the level at **maximum**. Click on the command button **Apply**.

4 - Select the ***Mesh*** page. Select the mesh of your choice in the drop-down list. Click on the command button **Apply**

5 - Select the ***Print detail level*** page. Fix the level at **minimum**. Click on the command button **Apply**

6 - Continue with the specification of parameters...

For this example, you will have in the display list :

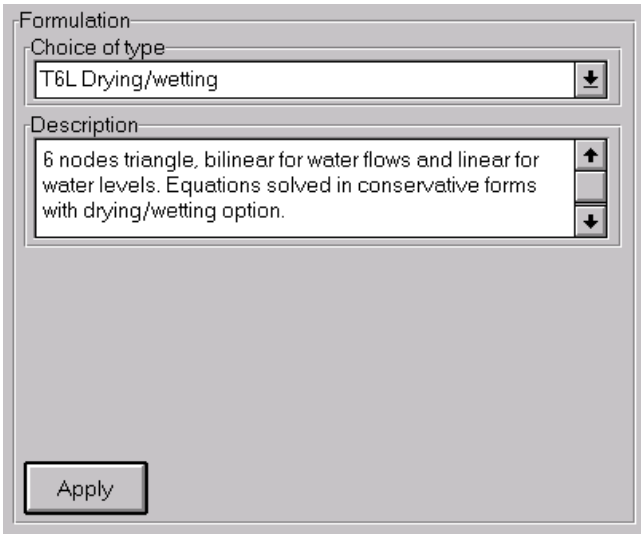
```
Print detail level : Minimum
Formulation : T6L Drying/wetting
Print detail level : Maximum
Mesh : Original mesh
Print detail level : Minimum
...
```

Notes :

- The default print level is the minimum print detail level.
- If you specify at a location a print detail level, this level will remain effective until the end of the commands, unless you specify another level further along.
- In many cases, the intermediate and maximum levels produce similar results.

Formulation

The formulation parameter is the first command required. It is a **prerequisite** for the other parameters.


Formulation
Choice of type
T6L Drying/wetting
Description
6 nodes triangle, bilinear for water flows and linear for water levels. Equations solved in conservative forms with drying/wetting option.
Apply

To specify the formulation :

1 - In the parameters zone, click on the formulation radio button. The page above is displayed in the right part of the parameters zone.

2 - Select the formulation in the drop-down list. The list contains all the formulations which are supported. At the time, only the formulation **T6L Drying/Wetting** is available.

3 - Click on the command button **Apply** at the bottom of this parameters page. The command is then added in the display list in the upper part of the simulation window.

Note :

- You can modify, remove or move this command later, see *Zone of specifications* display.

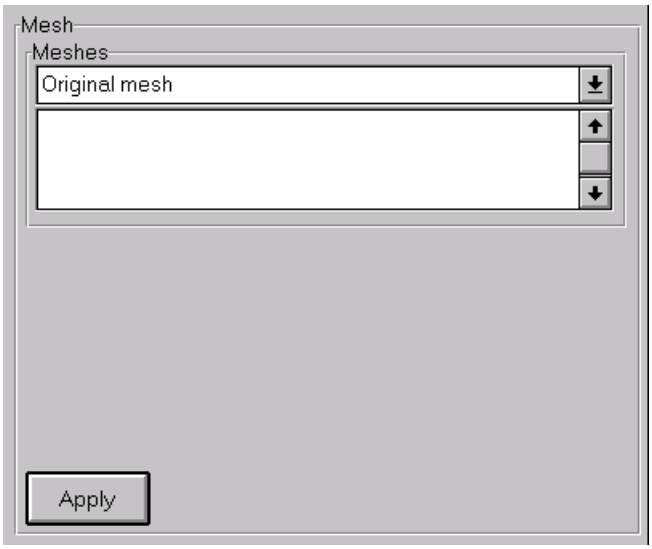
Mesh

In the mesh selection page, you can choose the mesh used by the simulator.

This mesh must have been generated by MODELEUR in the Mesh mode. (See the documentation on *Data preparation*) Also, this mesh must contain only ***T6L*** elements and be linked to a coordinate system of the type **cartographic** or **cartesian**, i.e. coordinates specified in a cartesian system (in meters or in analytical units). A **geographic** coordinate system, i.e. coordinates specified in longitude/latitude, would not be accepted by the simulator.

Prerequisite :

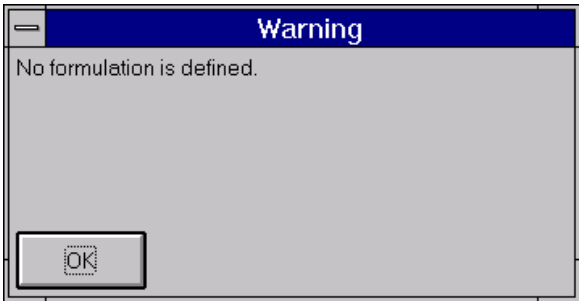
- The formulation must have been specified. (See *Formulation.*)



To specify the simulation mesh :

1 - In the parameters zone, select the mesh radio button. The page above is displayed in the right part of the parameters zone.

If the formulation has not been defined, a warning box appears. Click on OK and you will have to specify the formulation before proceeding any further.



2 - Select the mesh of your choice in the mesh drop-down list. The list contains all the meshes acceptable for the hydrodynamic simulation.

3 - Click on the command button **Apply** at the bottom of the parameters page. Your selection appears in the display list in the form of a mesh command.

Notes :

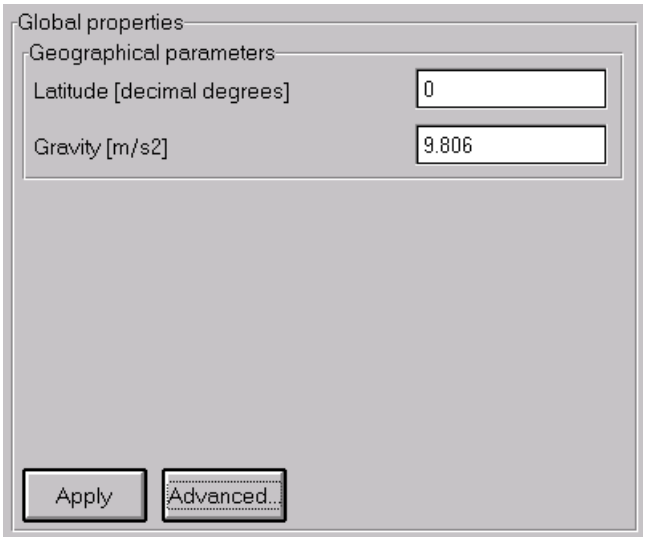
- The specification of a simulation mesh is a ***prerequisite*** for most of the parameters following.
- You can modify, remove or move this command later, see *Zone of specifications* display

Global properties

In the global properties specification page, you can specify several physical and numerical properties useful to the simulator.

However, default values are assigned and should be convenient for most simulations. This is why the global properties page is divided in two :

- Geographic parameters
- Advanced parameters.



To specify the global properties :

1 - Sin the parameters zone, select the ***Global properties*** radio button. The page above is displayed in the right part of the parameters zone..

2 - The geographic parameters are displayed directly in the page. Validate or specify these parameters. (See *Geographic parameters*.)

3 - If you wish, you can consult or specify the advanced parameters by clicking on the command button **Advanced** at the bottom of the global properties page.

4 - An **Advanced global properties** dialog box appears. You can specify these various data fields. (See *Advanced parameters*.)

5 - Click on **OK** in the Advanced global properties box to confirm your specifications or click on **Cancel** if you do not want to modify them.

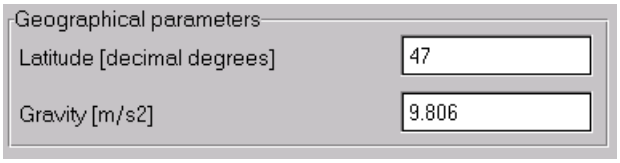
6 - Once everything is specified correctly, click on the command button **Apply** at the bottom of the ***Global properties*** page. A command is added in the simulation display list.

Note :

- You can modify, remove or move this command later, see *Zone of specifications* display

Geographic parameters

The page concerning the *geographic parameters* of the ***global properties*** contains two fields.



1 - The first field concerns the latitude at which the data are located. The latitude must be specified in decimal degrees, i.e. 47 degrees 30 minutes must be specified as 47.5 in decimal degrees.

2 - The second field concerns the altitude at which the data are located. It is specified as the gravity at this altitude in m/s^2 . The formula to obtain gravity depends on the latitude and the altitude of your data. The interval of acceptable values is [9.75258282, 9.86605398].

The formula used to calculate gravity is :

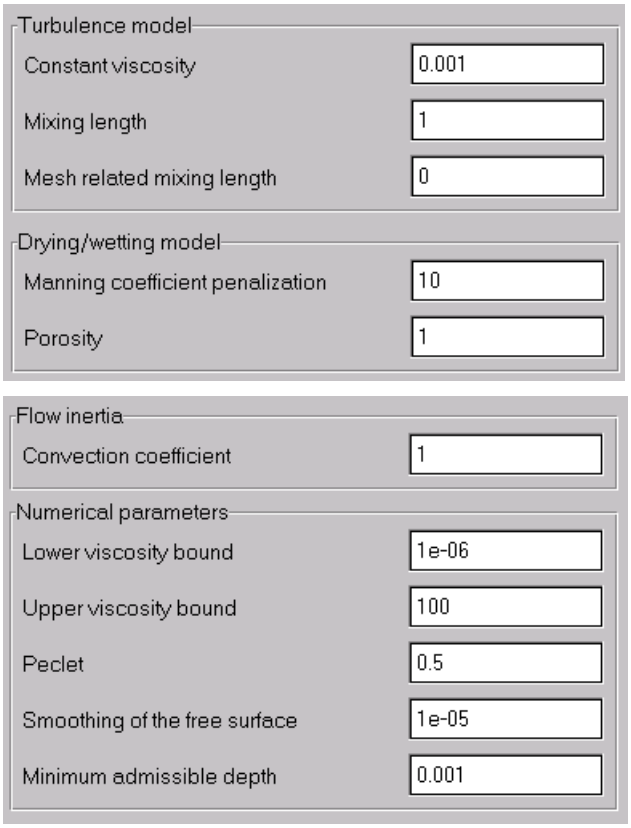
Φ = latitude
Z = altitude from sea level
domain Z = [-11000, 9000] where the minimum is the Mariannes Deep and the maximum is the Everest.

$$g = 9.78031 * (1 + 0.005301 \sin^2(\Phi) - 0.000006 \sin^2(2\Phi) - 0.000000315*Z)$$

3 - If you do not want to modify the advanced parameters, click on the command button **Apply** at the bottom of the global properties page. The command is added.

Advanced parameters

The advanced parameters section is accessible by clicking on the command button **Advanced...** at the bottom of the **global properties** page. A dialog box appears and contains four large zones of data.



1 - The turbulence model contains three fields : constant viscosity, mixing length and mixing length related to the dimension of the mesh of your simulation mesh.

2 - The drying/wetting model contains two fields : Manning penalization on drying and porosity.

3 - The flow inertia contains one field : the convection coefficient.

4 - The numerical parameters contain five fields : the lower and the upper bounds, the Peclet coefficient, the free surface smoothing coefficient, and the minimum depth admissible.

5 - Click on **OK** to confirm your modifications or on **Cancel** to cancel your modifications.

Note :

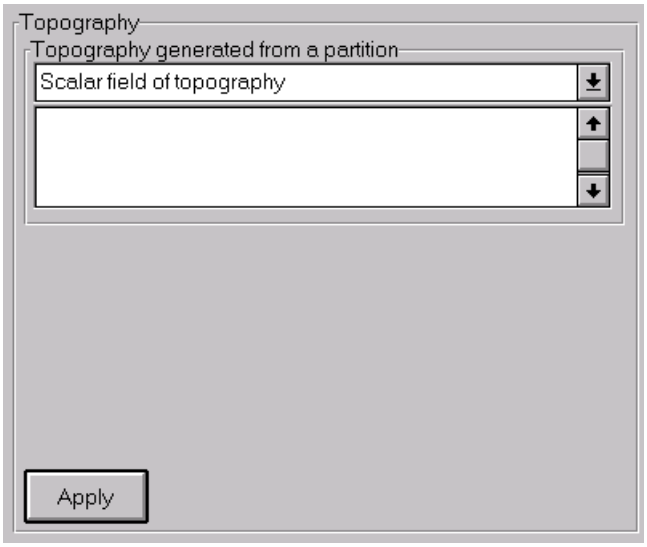
- For more information, refer to the documentation on the HYDROSIM simulator.

Topography

In the topography specification page, you can select the topography that will be used by the simulator. The topography must necessarily be compatible with the mesh selected.

Prerequisite :

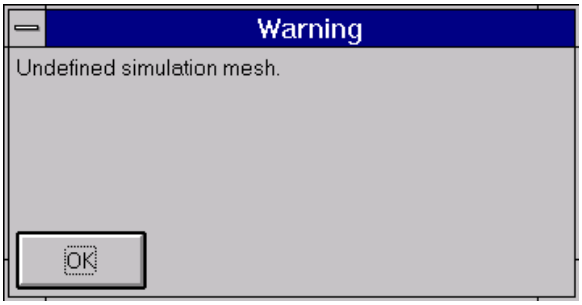
- The mesh must have been specified before the topography is specified. (See *Mesh*.)



To specify the topography to use :

1 - In the parameters zone, select the **Topography** radio button. The topography page above is displayed in the right part of the simulation parameters zone.

2 - If the simulation mesh has not been specified, a warning message is displayed. Click on OK to close the warning box ; you must then specify a simulation mesh before proceeding any further (See *Mesh*.)



3 - In the drop-down list, select the topography of your choice. The lists contains only the **scalar fields - topography** linked to the simulation **mesh** previously selected.

4 - Click on the command button **Apply** to confirm your selection. A command is added to the simulation display list.

Note :

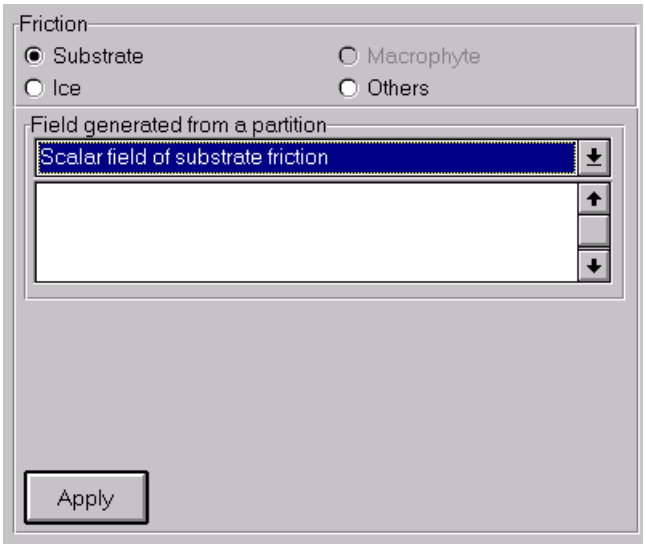
- You can modify, remove or move this command later, see *Zone of specifications* display.

Friction

In the friction specification page, you can add to the terrain model the friction coefficients that you have specified. There are several friction coefficients admissible for the simulator : substrate, ice, macrophytes (not available) and the friction directly specified as Manning coefficients.

Prerequisite :

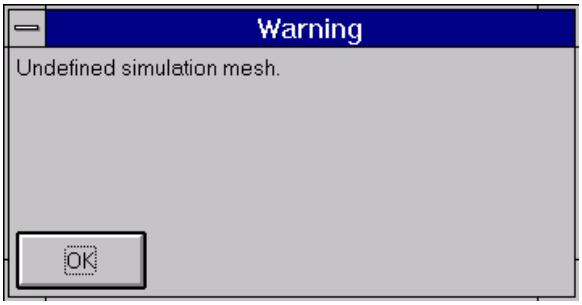
- The mesh must have been specified before the friction is specified. (See *Mesh*.)



To specify a friction coefficient :

1 - In the parameters zone, select the **Friction** radio button. The friction specification page above appears.

2 - If the simulation mesh has not been specified, a warning message is displayed. Click on OK to close the warning box ; you must then specify a simulation mesh before proceeding any further (See *Mesh*.)



3 - Select the type of friction that you want to specify in the upper part of the friction page. You have the choice between substrate, ice and the other friction (Manning).

4 - In the drop-down list, select the friction field of your choice. The list contains all the friction coefficients generated on the simulation mesh selected for the type of friction selected. For

substrate, you have all the **scalar fields - substrate friction**. For ice, you have all the **fields - ice** and for the other friction coefficients, you have all the **scalar fields - Manning friction**.

5 - Click on the command button **Apply** at the bottom of the friction page. A command is added in the simulation display list.

6 - Repeat steps 1-5 for all the types of friction that you wish to add.

Notes :

- You can modify, remove or move this command later, see *Zone of specifications* display.
- The friction coefficients will be cumulated to form a single Manning friction coefficient. The formula is :

  $coeff = sqrt((frot_1)^2 + ... + (frot_N)^2)$
  where **sqrt** is the square root.

- The field - ice will be transformed as follows. The component friction will be taken into account in the preceding formula. The second component (ice thickness) will be kept and considered separately by the simulator.

Initial conditions

In the initial conditions specification page, you can specify to the simulator in which conditions it will launch the simulation. There are two options :

- Initial conditions without data
- Initial conditions with data.

To specify the initial conditions :

1 - In the parameters zone, select the **Initial conditions** radio button. The initial conditions page appears at the right of the parameters zone.

2 - Select the type of initial conditions by clicking on the appropriate radio button.

3 - if you have selected the initial conditions **without** data, see *Initial conditions without data*. if you have selected the initial conditions **with** data, see *Initial conditions without data*.

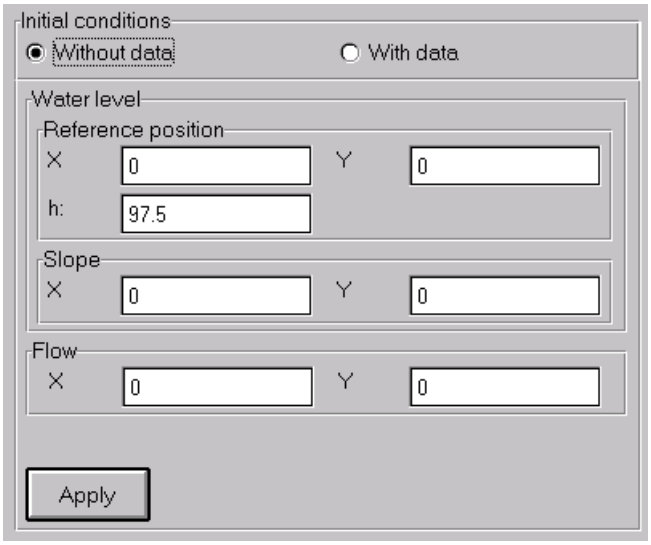
4 - Click on the command button **Apply** at the bottom of the initial conditions page. The command is added in the simulation display list.

Note :

- You can modify, remove or move this command later, see *Zone of specifications* display.

Initial conditions without data

In the initial conditions without data page, you can enter the initial conditions allowing to launch the simulator in coherent conditions without data..



To specify the initial conditions without data :

1 - First, you must determine the parameters dictating the initial water level.

The first fields concern the reference position. You must specify a position in X and in Y , as well as an initial water level. Generally, the level is set at the average of the minimum and maximum levels of your boundary conditions.

You must then specify the slope imposed on this water level in X and in Y.

2 - The discharge must then be fixed in X and in Y.

3 - For more information on these criteria, se the documentation on the simulator.

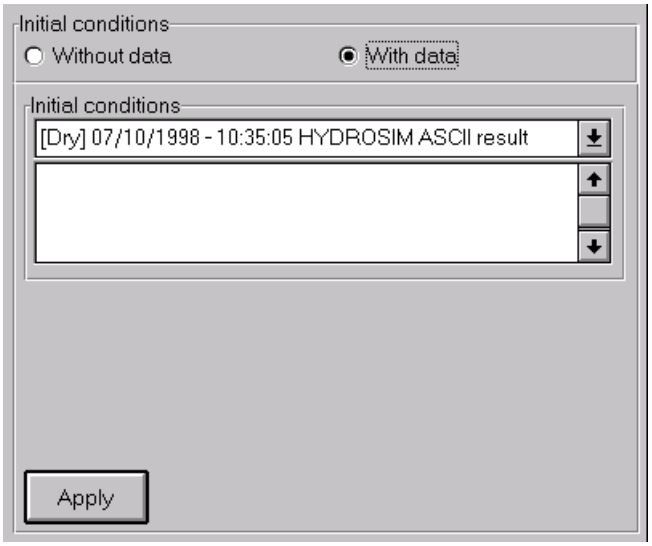
4 - Click on the command button **Apply** to confirm your specifications. The command is added to the simulation display list.

Initial conditions with data

In the initial conditions with data page, you provide the simulator with a solution already generated by the simulator and ask to restart with this solution.

Prerequisite :

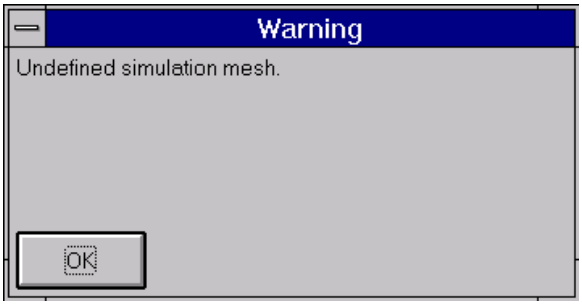
- The mesh must have been specified before the initial conditions with data are specified. (See *Mesh*).



To specify initial conditions with data :

1 - Select in the drop-down list the initial data for HYDROSIM. The list contains the initial solutions available on the simulation mesh selected.

2 - If the simulation mesh has not been specified, a warning message is displayed. Click on OK to close the warning box ; you must then specify a simulation mesh before proceeding any further (See *Mesh*.)



3 - Click on the command button **Apply** at the bottom of the initial conditions with data page. The command is added to the simulation display list.

Notes :

- The initial conditions for HYDROSIM are obtained after having launched a simulation and asked for the HYDROSIM ASCII results in the section **Results**. (See *Results*.)

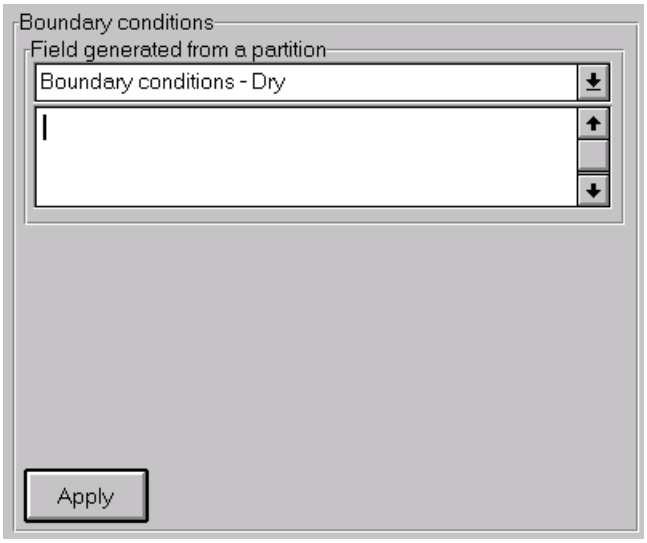
- It is possible to launch a simulation with a very rough mesh to obtain water levels quickly. Then, you can re-launch a simulation with a more refined mesh to continue the processing. To execute this type of processing, you must **project** the result obtained with the rough mesh on a more refined mesh. (See the documentation on *Using the tools - Projection*.)

Boundary conditions

In the boundary conditions page, you can specify the boundary conditions of the simulation by selecting a precise hydrological event. The boundary conditions must have been specified and generated on the simulation mesh selected.

Prerequisite :

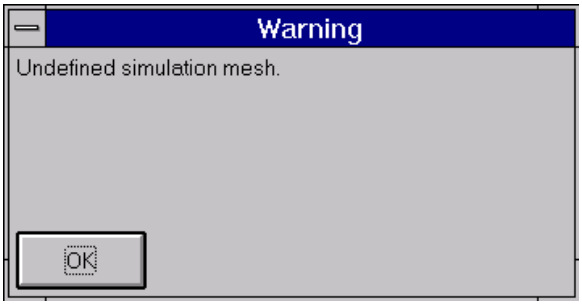
- The mesh must have been specified before the boundary conditions are specified. (See *Mesh*.)



To specify the boundary conditions :

1 - In the parameters zone, click on the **Boundary conditions** radio button. The boundary conditions page above appears.

2 - If the simulation mesh has not been specified, a warning message is displayed. Click on OK to close the warning box ; you must then specify a simulation mesh before proceeding any further (See *Mesh*.)



3 - In the drop-down list, select the boundary conditions of your choice. The list contains all the boundary conditions generated on the simulation mesh selected.

4 - Click on the command button **Apply** at the bottom of the boundary conditions page. The command is added to the simulation display list.

Note :

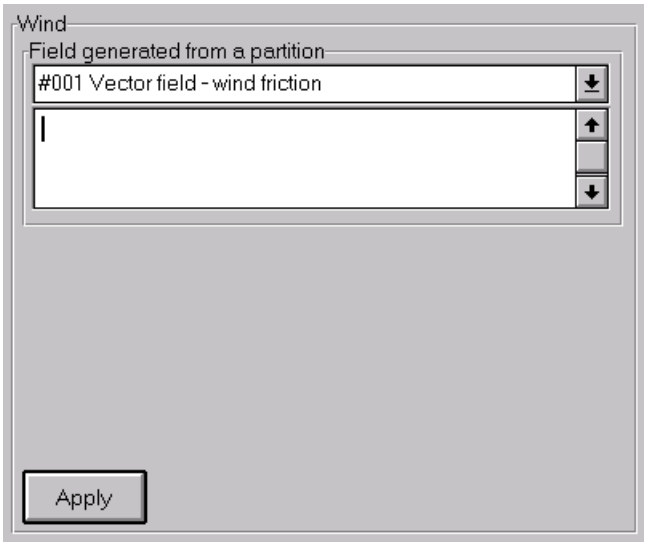
- You can modify, remove or move this command later, see *Zone of specifications* display.

Wind

In the wind parameters page, you can specify the constraints related to the wind by selecting a **vector field - wind** specified in the **Modeleur**, **Wind** mode.

Prerequisite :

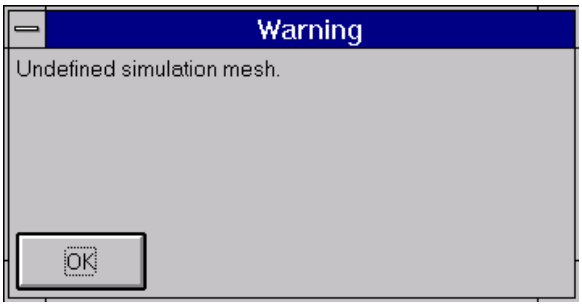
- The mesh must have been specified before the wind is specified. (See *Mesh*.)



To specify the wind parameters :

1 - In the parameters zone, select the **Wind** radio button. The wind parameter page above is displayed.

2 - If the simulation mesh has not been specified, a warning message is displayed. Click on OK to close the warning box ; you must then specify a simulation mesh before proceeding any further (See *Mesh*.)



3 - In the drop-down list, select the **vector field - wind** of your choice. The list contains all the vector field - wind generated on the simulation mesh selected.

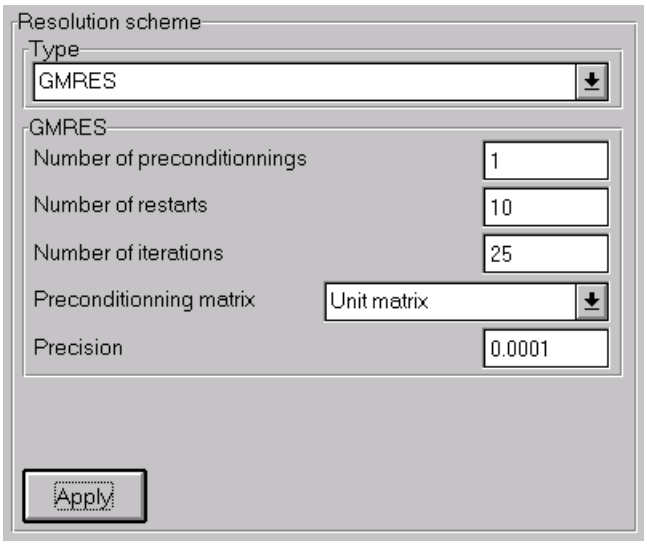
4 - Click on the command button **Apply** at the bottom of the wind parameters page. The command is added to the simulation display list.

Note :

- You can modify, remove or move this command later, see *Zone of specifications* display.

Resolution scheme

In the resolution scheme page, you can specify the resolution scheme to use with the simulator.



To specify the resolution scheme :

1 - In the parameters zone, select the **Resolution scheme** radio button. The page above appears.

2 - In the drop-down list, select the resolution scheme of your choice. At this time, the list contains only GMRES.

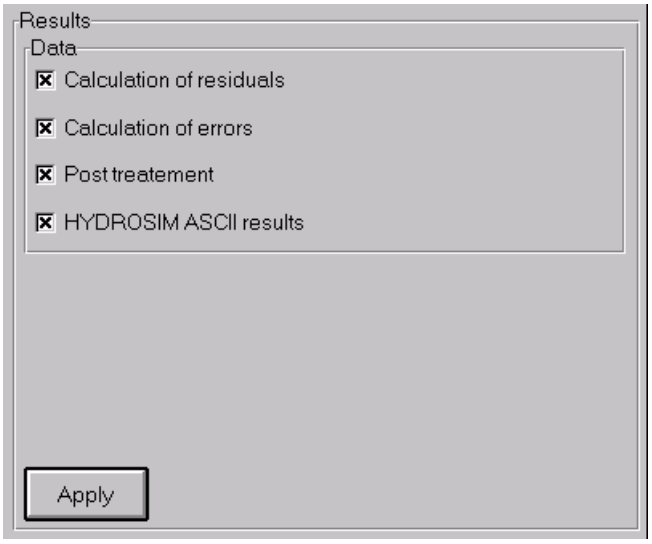
3 - Enter the parameters of the resolution scheme selected. For GMRES, these parameters are the preconditioning matrix and four numerical fields : the number of preconditioning, the number of restart, the number of iterations and the precision which defines the convergence criteria of the simulation.

4 - For more information on these parameters, consult the documentation on the simulator.

5 - Click on the command button **Apply** at the bottom of the page. The command is added to the simulation display list.

Results

In the results page, you can specify what you want to obtain as results of the simulation.



To specify the results to obtain :

1 - In the parameters zone, select the **Results** radio button. The page above appears.

2 - Select the check boxes that interest you.

3 - The calculation of residuals helps you validate the value of your result : residuals on velocity in X and Y, and residuals on depth.

4 - The calculation of errors also helps you validate the value of your result : error on velocity in X and Y and error on depth.

5 - Post-processing contains all the results interesting in terms of data : velocity, surface level, depth, Froude number, shear stress velocity, local and global mass conservation, numerical and physical viscosity.

6 - The HYDROSIM ASCII result will help you launch another simulation with this result. You can also project this result toward another compatible mesh to build a simulation starting from this result.

7 - Click on the command button **Apply** at the bottom of the page. The command is added to the simulation display list.

Launch and monitor a simulation

Once you have completed the construction of your simulation, you are now ready to launch the simulation and, if wanted, to monitor it.

Click on the command button **Launch** in the tool bar at the left of the simulation window. The launch window appears.

You have various options to launch the simulator :

1 - Launch and monitor the simulation

In this case, the software generates all the data needed by the simulator. Then, it will launch the simulator and enter in a waiting loop to monitor the simulation. At this point, the software only displays the progress of the simulation and you cannot do other work.

2 - Launch the simulator but do not monitor the simulation

In this case, the software generates all the data needed by the simulator. Then, it will launch the simulator. You can keep working.

3 - Launch the simulator later.

In this case, the software generates all the data needed by the simulator. You will later have to manually launch the simulator by clicking on the script **`goexe.bat`** in the simulation directory.

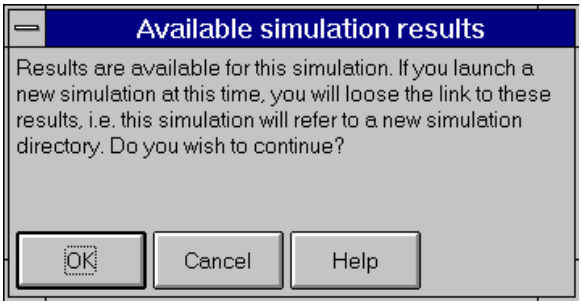
Click on **OK** to launch the simulator or on **Cancel** to cancel the operation.

Notes :

- When you launch the simulator, you will see several console windows appear. They are exporting the data to the simulation directory. If you have asked for the monitoring, refocus on the simulation window so that you can monitor the progress at the bottom left of the window.
- You can consult the simulator report by clicking on the command button **Reports** at the left of the simulation window. The simulator report will be displayed in a text editor.

Restriction :

- Every time you ask to launch a simulation, a new simulation directory is created. If you have not recovered the results of the preceding simulation, if there is one, you loose the link to it.
- You will be warned of this situation when launching.


Available simulation results
Results are available for this simulation. If you launch a new simulation at this time, you will loose the link to these results, i.e. this simulation will refer to a new simulation directory. Do you wish to continue?
OK
Cancel
Help

Click on **OK** to continue or click on **Cancel** to cancel the launch. You can then recover the results before launching another simulation.

Recover the results

When the simulation is completed, you must recover the results. These results correspond to what has been asked in the results command. (See *Results*.)

To recover the results of the simulation, click on the command button **Results** in the tool bar at the left of the simulation window.

Immediately, all the results requested will be imported in your project ; you will again see console windows appear. If you requested the totality of results, here is what may be re-imported :

[Dry] 07/10/1998 - 10:34:57 Residual discharge in X
[Dry] 07/10/1998 - 10:34:58 Residual discharge in Y
[Dry] 07/10/1998 - 10:34:58 Residual water level
[Dry] 07/10/1998 - 10:34:59 Error discharge in X
[Dry] 07/10/1998 - 10:34:59 Error discharge in Y
[Dry] 07/10/1998 - 10:35:00 Error water level
[Dry] 07/10/1998 - 10:35:00 Velocity modulus
[Dry] 07/10/1998 - 10:35:00 Velocity vector
[Dry] 07/10/1998 - 10:35:01 Water level
[Dry] 07/10/1998 - 10:35:02 Depth
[Dry] 07/10/1998 - 10:35:02 Shear stress velocity
[Dry] 07/10/1998 - 10:35:03 Froude number
[Dry] 07/10/1998 - 10:35:03 Local mass balance
[Dry] 07/10/1998 - 10:35:04 Numerical viscosity
[Dry] 07/10/1998 - 10:35:04 Physical viscosity
[Dry] 07/10/1998 - 10:35:05 Global mass balance
[Dry] 07/10/1998 - 10:35:05 HYDROSIM ASCII result

The labels generated are all prefixed with the date and time to make search easier.

Validate the results obtained

The results obtained from your simulation are not automatically valid. You must validate and calibrate them through an iterative process.

Consult the documentation on the simulator to find the important aspects to validate and how to calibrate.

Content of a simulation directory

When you launch a simulation, a directory is created to receive all the files necessary for the execution of this simulation by the simulator. This section contains the description of the files generated in this directory.

`command.inp`

Input commands of the simulation

`exe.bat and geoexe.bat`

Manage the simulation

`post.bat and gopost.bat`

Manage the post-processing

`pre.bat and gopre.bat`

Manage the pre-processing

`hyprno0.prn`

Nodal properties

`import.bat`

Importation of the results in MODELEUR

`progress.dat`

Progress status of the simulation

`result.out`

Results of the simulation

`simul000.bth`

Topography

`simul000.cnd`

`Boundary conditions`

`simul000.cor`

Coordinates of the nodes of the hydrodynamic mesh

`simul000.ele`

Connectivities of the elements of the hydrodynamic mesh

`simul000.err`

Errors on degrees of freedom

`simul000.fin`

ASCII file of the simulation results

`simul000.frs`

Substrate friction

`simul000.pst`

File containing all the results of the post-processing

`simul000.rep`

Coordinate system

`simul000.res`

Residuals on the degree of freedom

`simul000.slc`

Concentrated solicitations

`simul000.vno to simul015.vno`

Files of the residuals, the errors and the post-processing. These are files created when recovering the results of the simulation. The data they contain can be displayed in MODELEUR.

Chapter 8 Using the tools

The software offers a wide array of integrated tools to complete your work of fluvial analysis.

You have access to a Data Probe with which you can get positions, and values from scalar and vector fields.

You also have access to a programmable calculator adapted to the handling of scalar and vector fields. You have a programming language to write functions compatible with the calculator.

You also have a projection tool to project a field created on a mesh toward another mesh, thus creating a new field.

Finally, you have a profile design tool with which you can specify a transect using a profile partition. This partition will then be used in the static display mode to display profiles on a given field.

Here are the various tools at your disposition :

- Data probe
- Calculator
- Function editor
- Projection tool
- Profile mode.

Data probe

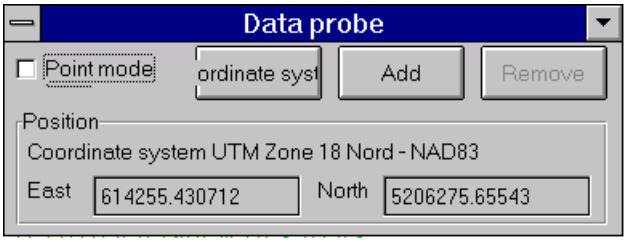
The data probe is an extremely useful tool to view interactively values from scalar and vector fields.

The probe offers the possibility to view the coordinates according to the position of the mouse in a display window.

To access this tool, select the item **Probe** in the **Tool** menu. A pop-up window appears.

Prerequisite :

- A project must be open (See the documentation on Project managementAID_PRJ_CHARGE@gestproj.hlp)
- A display window must be open. (See the documentation on Display controlAID_MNU_FEN_CREE@ctlaffic.hlp)



When you move the mouse over a display window (click on the title bar to focus on it), the corresponding position in terms of spatial location is displayed in the fields of the group **Position** of the **Data probe** window.

You can select the type of coordinate system used for the position of the probe. (See *Select the type of coordinate system*.)

You can switch to the **On- click mode** or to the **Continuous mode** if you wish by clicking on the check box in the Probe window. (See *On-click - continuous mode*.)

If you want to consult the value of a field at any point in the domain, you can do it for every scalar and vector fields present in your project. (See Add a field to the data probe and Remove a field from the data probe.)

Select the type of coordinate system

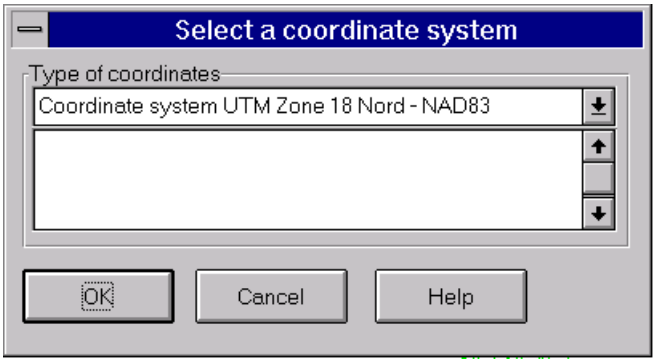
You can choose the type of coordinate system to be used in the display of positions in the probe.

Prerequisite :

- The **Probe** pop-up window must be open.

To select a coordinate system, follow these steps :

1 - Click on the command button **Coordinate system** of the probe. A coordinate system selection box appears.



2 - In the drop-down list, select the coordinate system of your choice. The list contains all the coordinate systems available in your project. In the case of a georeferenced project, you

have cartographic coordinate systems (UTM, MTM, ...) and geographic coordinate systems (longitude/latitude).

3 - Click on **OK** to confirm the selection or on **Cancel** to keep the current selection.

4 - The name of the new coordinate system will be displayed in the group **Position** of the probe.

Notes :

- The appearance and the content of the group **Position** depends on the type of coordinate system used.
- Thus, if the coordinate system selected is cartesian, the position values ill be [X, Y] in the abstract unit of the project.
- If you select a cartographic coordinate system , the position values will be [Easting, Northing] in meters and the values as a function of the type of cartographic data.
- If you select a geographic coordinate system, the position values will be [Longitude, Latitude] in degrees, minutes, seconds, decimals. For example, 73d29'45.3827"W means 73 degrees, 29 minutes, 45.3827 seconds West.

On-click - continuous mode

You can switch from a continuous display mode to a on-click display mode (and conversely).

The **continuous mode** means that the probe displays in continuous the information sent by the system when the mouse moves over a display window.

The **on-click mode** means that the probe only displays the information when you click in the display window.

Prerequisite :

- The **Data probe** pop-up window must be open.

To switch between modes, click on the check box **On-click mode** of the probe window. If the option is checked, you are in the **on-click mode,** otherwise you are in the **continuous mode**.

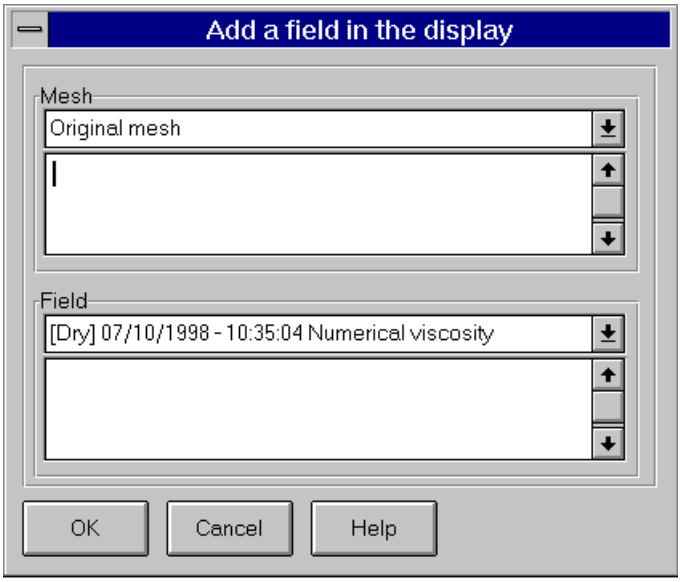
Add a field to the data probe

You can add a scalar or a vector field to view the values in the probe.

- coordinate system

To add a field in the probe, follow these steps :

1 - Click on the command button **Add** of the probe window. A field selection dialog box appears.



2 - You must first the mesh on which the field wanted has been generated. In the drop-down list of **Meshes**, click on the mesh of your choice. The list contains all the meshes available in your project.

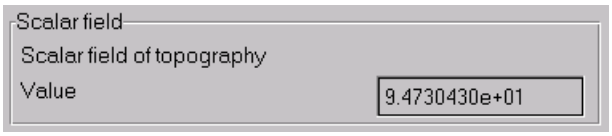
3 - In the drop-down list of **Fields**, select the field that you want to add in the probe. The list contains all the scalar and vector fields generated on the mesh selected.

4 - Click on **OK** to add the field selected or on **Cancel** if you do not want to add it to the probe.

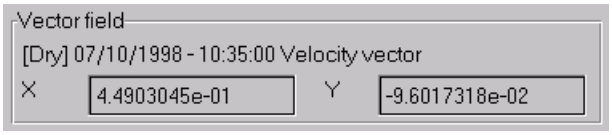
5 - The probe now contains a new section corresponding to the field added. The appearance of the group depends on the type of field selected. The title of the group will be either **Scalar field** or **Vector field**.

Notes :

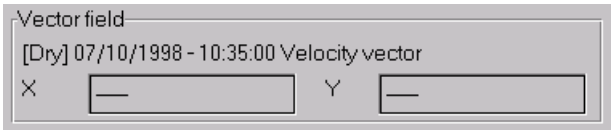
- The group Scalar field contains the name of the scalar field and the value..



- The group Vector field contains the name of the vector field and the values in X and in Y.



- When the mouse moves in a display window, the values of these fields are displayed in the appropriate locations. If there is no value defined at the mouse position for a given field, dash lines are displayed instead of values.



- You can add as many fields as you wish, the limit being the capacity of the interface to display them on the screen.

Remove a field from the data probe

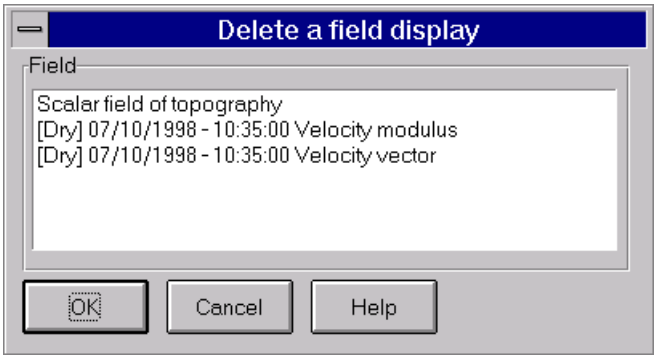
You can remove from the probe a field previously added.

Prerequisite :

- The **Data probe** pop-up window must be open.

To remove a field from the probe, follow these steps :

1 - Click on the command button **Remove** in the probe window. A Remove a field dialog box appears.



2 - In the drop-down list, select the field that you wish to remove from the probe. The list contains all the fields currently added in the probe.

3 - Click on **OK** to remove the field or on **Cancel** if you do not want to remove it.

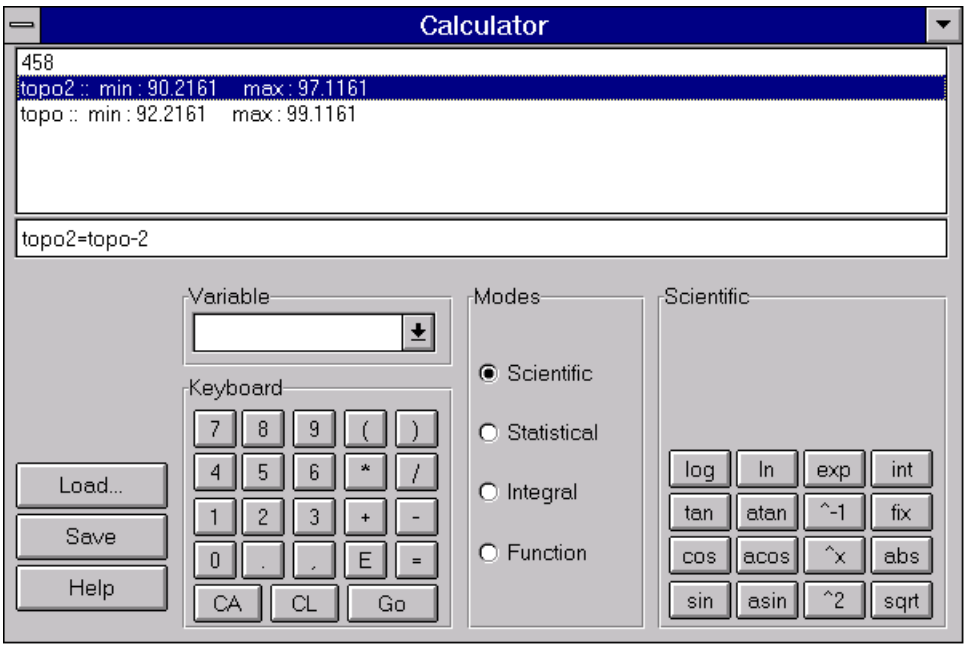
4 - The group corresponding to the field is then removed from the probe.

Calculator

The calculator is a calculation tool which differs from traditional calculators by its capacity to work on scalar and vector fields with the database. Thus, you can create new data derived from existing entities.

This is made possible by its evolved programming language which allows to write functions compatible with the calculator.

To accede to the calculator, select the item **Calculator** in the **Tool** menu. A pop-up window appears.



The calculator has several zones with which you can interact.

1 - At the top of the window, you will find the calculation results section. Following the execution of a command, the result appears in this list. (See *Results zone*.)

2 - Below the results zone, you have the command line from which you specify the command to be executed. (See *Command line*.)

3 - In the left part of the window, you have the buttons zone to load and save scalar and vector fields. (See *Load a field* and *Save a field*.)

4 - You then have the numerical keyboard zone to specify the commands. You can also choose a variable in the drop-down list of variables. (See *Numerical keyboard* and *List* of variables.)

5 - In the right part of the window, you have the pages for predefined functions and user functions. (See *The functions of the calculator*.)

See also:

- See the documentation on the programming languageAID_LANGAGE_PROGRAMMATION@langprog.hlp
- See Function editor.

Results zone

The result zone is a list in Which the calculator inserts on one line at a time the results of a command. (See the section *Command line* to find out how to write a command).

The display offered as results depends on the type of result. A numerical results simply displays the number (**scalar** and **integer**). The same applies for the booleans and the strings of characters (**logical** and **string**). For the type **vector**, the values in X and Y are displayed. Finally, The scalar and vector fields display their min/max (**scalarfield** and **vectorfield**).

When you execute a command, the result in displayed on the first line of the list of results. To display on the command line the command that produced the result, select the item in the list of results. The command will appear on the command line.

Command line

The command line is where you can specify any command.

You can use the keyboard directly to enter the command, or use the calculator buttons, either on the numerical keyboard or the function buttons.

To execute the command, either click on **Enter** on the keyboard or on **Go** on the numerical keyboard of the calculator.

Numerical keyboard

The numerical keyboard zone of the calculator allows you to control the composition and the execution of the commands.

The keyboard has the traditional number keys (**0**..**9**), the numerical dot (**.**), the comma (**,**), the operators **(+**, **-**, ***, /**, **=**), the parentheses (**(**, **)**) and the exponential symbol (**E**) for the scientific notion.

When you click on one of these keys, the corresponding value is displayed on the command line at the position of the cursor.

The **CA** key erases completely the list of results and frees the display capacity in this zone.

The **CL** key erases completely the command line and frees the display capacity in this zone.

Finally, the **Go** key asks for the execution of the command displayed on the command line.

List of variables

The variables drop-down list contains the list of all the variables available. In the drop-don list, select the variable of your choice and it will be inserted on the command line at the position of the cursor.

you have many ways to add a variable to this list. When you load a field into the calculator, you give a name to the field and it is added to the list of variables. (See *Load a field*). If you give a name to a command, i.e. if you ask to include in a variable the result of the command, it will also be added to the list of variables.

Example:

```
result = 2 * (3 + 4)
```

The variable **result** is created and added in the list of variables.

The functions of the calculator

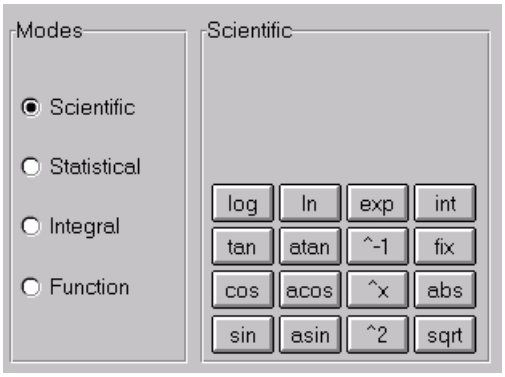
The functions of the calculator are distributed in four distinct groups :

- Scientific mode
- Statistical mode
- Integral mode
- User functions mode

The radio buttons of the section **Modes** located immediately to the right of the keyboard allow the selection of one of these groups. The functions of the group selected become accessible in the right part of the calculator.

Scientific mode

The page of scientific functions provides the use of predefined functions in the construction of a command. By simply clicking on one of the buttons, the function is added to the command line at the position of the cursor.



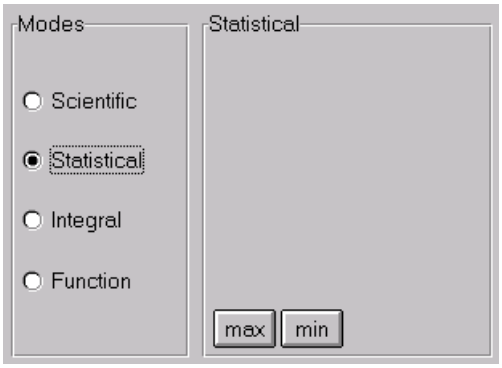
The following functions are available in this page:

- General functions : **abs**, **int**, **fix**, **sqrt**
- Trigonometric functions: **sin**, **cos**, **tan**, **asin**, **acos**, **atan**
- Logarithmic and exponential functions: **log**, **ln**, **exp**
- Other operators: **^2**, **^x**, **^-1**.

Consult the documentation on the programming languageAID_LANGAGE_PROGRAMMATION@langprog.hlp to find the detailed meaning and the limits of applicability.

Statistical mode

The page of statistical functions provides the use of predefined functions in the construction of a command. By simply clicking on one of the buttons, the function is added to the command line at the position of the cursor



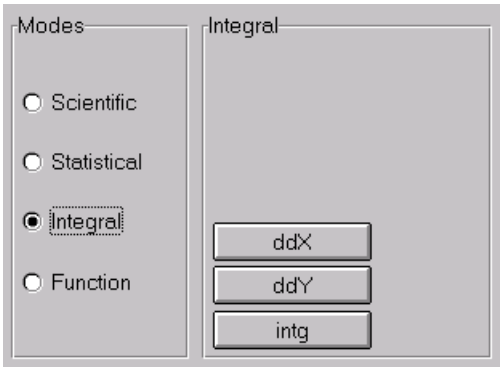
The following functions are available in this page:

- General functions: **max**, **min**

Consult the documentation on the programming languageAID_LANGAGE_PROGRAMMATION@langprog.hlp to find the detailed meaning and the limits of applicability.

Integral mode

The page of integral functions provides the use of predefined functions in the construction of a command. By simply clicking on one of the buttons, the function is added to the command line at the position of the cursor



The following functions are available in this page:

- General functions: **ddX**, **ddY**, **intg**.

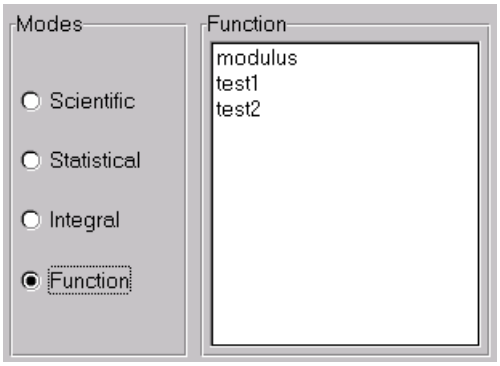
Consult the documentation on the programming languageAID_LANGAGE_PROGRAMMATION@langprog.hlp to find the detailed meaning and the limits of applicability.

User functions mode

You can define and apply user functions. They must be written in the programming language of the calculator and must be edited

in the function editor available in the software. (See *Function editor*).

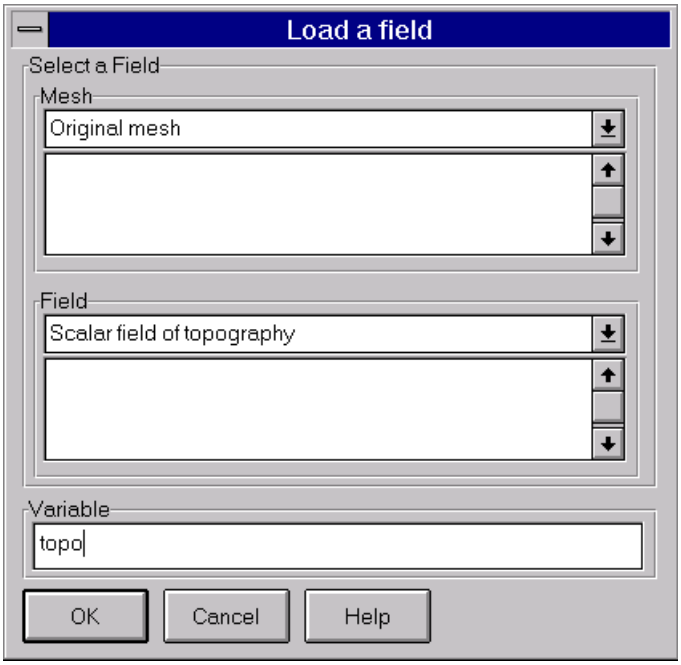
In this page, you will find all the functions available in your project. By simply double-clicking on a user function, the function is added to the command line at the position of the cursor.



Load a field

Through variables of the types **scalarfield** and **vectorfield**, the calculator provides the possibility to execute operations on scalar and vector fields of the current project. To execute operations on such fields, they must first have been loaded in appropriate variables :

1 - Click on the command button **Load**. The **Load a field** dialog box appears.



2 - In the drop-down list of the section **Mesh**, select the mesh used to generate the field that you wish to load.

3 - In the drop-down list of the section **Field**, select the field that you wish to load.

4 - in the section **Variable**, enter the name of the variable which will represent the field in the calculator.

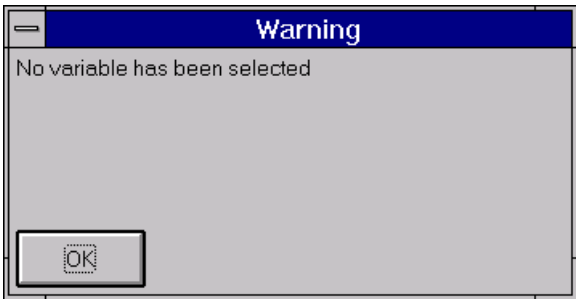
5 - Click on **OK**. The variable is added to the drop-down list of the section **Variable** of the calculator.

Save a field

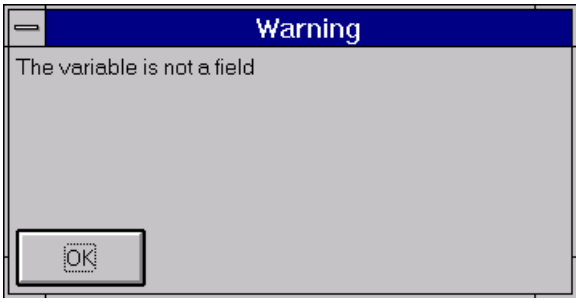
After generating data, you may want to save a scalar or vector field in the database. To save a field :

1 - In the variables drop-down list, select the variable of your choice.

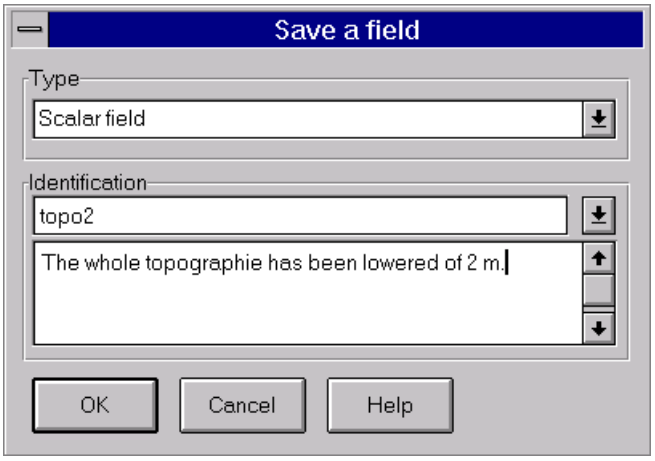
2 - Click on the command button **Save** at the left in the calculator window. If no variable has been selected, you will be asked to select one in the variables drop-down list.



3 - If the variable selected is not a scalar or a vector field, you will be warned of the situation.



4 - A Save a field dialog box appears.



5 - You can select the type of field in Which you want to save the data. If it is a **scalar field**, you will have access to all the types of scalar fields in the system. If it is a **vector field**, you will have access to all the types of vector fields in the system. Enter an appropriate name and description.

6 - Click on **OK** to save the new entity or on **Cancel** if you do not want to save it.

Function editor

The function editor provides the possibility to program customized functions to be used with the calculator.

The function editor is accessible in the **Tools** menu, item **Function editor**. The function edition pop-up window appears. It has two large zones :

1 - The edition zone.

2 - The function buttons zone.

The edition zone allows to view and edit the source code of the function.

The function buttons zone gives access to the various edition functions :

- Create a new function
- Load an existing function
- Save a function
- Save As... a function
- Close a function
- Delete a function
- Compile a function

Using combinations of keys on the keyboard, you can also:

- Cut selected code
- Copy selected code
- Paste code

See also:

- Calculator
- Documentation on the programming languageAID_LANGAGE_PROGRAMMATION@langprog.hlp

Create a new function

To create a new function, simply click on the function button **New**.

Immediately, code lines appear from the editor. A commentary structure for the documentation of the function you are about to write is proposed. An embryo of function body is also proposed.

You must adapt this code to answer your needs. You must determine the type of return (`return_type`), the name of the function (`function_name`), as well as the function parameters (`parameters`).

Notes:

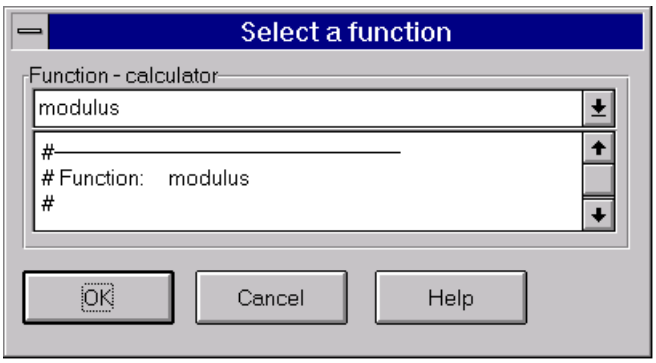
- The name of the function will automatically determine the label of the information related to this function. The name and the label must always be synchronized. This is why you will not have to enter a label and a description when you will save the function. The label will be the name of the function, and the description will be the commentary block above the code.

See also:

- Save a function
- Save As... a function.

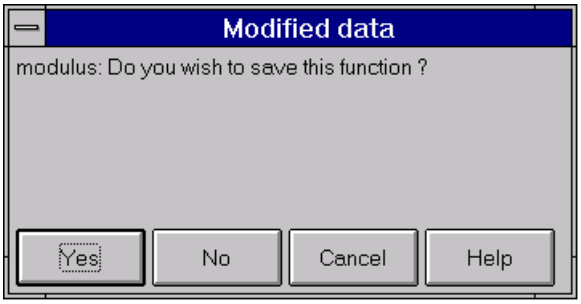
Load an existing function

To load an existing function, click on the function button **Load** at the bottom of the function edition window. The Load a function dialog box appears.



In the drop-down list, select the function of your choice. The list contains all the functions available in your project. The label is the name of the function, and the description contains the commentary block of this function.

Click on **OK** to load the function or on **Cancel** to cancel the operation. If a function is already activated and has been modified, the software will ask you if you want to save the modifications before closing it.



Click on the command button **Yes** to save or on **No** if you do not want to save the modifications. Click on **Cancel** if you do not want to load the function selected, and keep the modified function.

Save a function

To save a function under edition, click on the command button **Save**. An hourglass appears and when it disappears, the function has been saved. The function will not be marked modified anymore, if it was the case.

The save function does not ask you to provide a label and a description. The software uses the name of the function in the code and the commentary block as label and description respectively.

Warning:

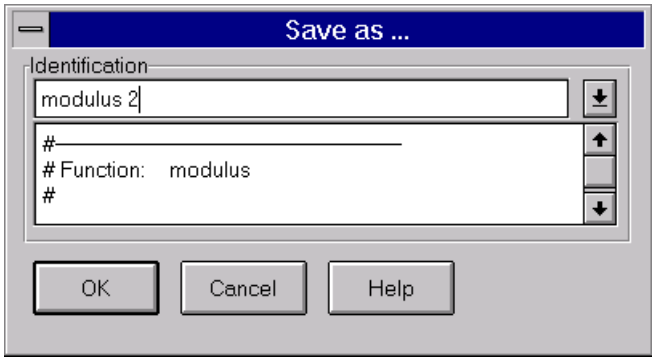
- Although you can modify the label and the description of functions in the entity manager, it is not recommended to do

so. It would make the function unusable by the motor of the calculator.

Save As... a function

To save a function as, you must have previously loaded a function. (See *Load an existing function*.)

Click on the command button **Save As...**. The Save As... dialog box appears.

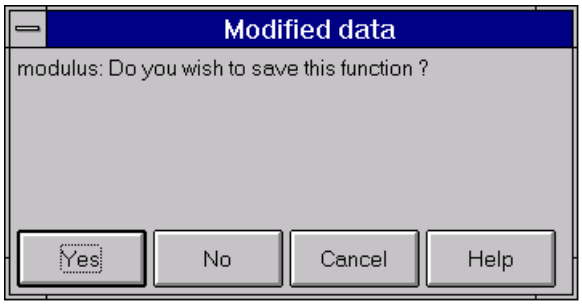


Give a unique label to your new function. The label will become the name of the function as such. Click on **OK** to confirm the save operation or on **Cancel** to cancel the operation.

You will notice that the function in the editor will automatically take the name given, and, in fact, the function under edition becomes this new function.

Close a function

To close a function, click on the function button **Close**. If the function is modified, a dialog box will inform you of the situation and you will be asked to save the modifications.



Click on the command button **Yes** to save, **No** to not save and close, or **Cancel** if you do not want to close the function.

If you proceed with the closure, there is no function left in the editor.

Delete a function

To delete a function, you must first have loaded it in the editor. (See *Load an existing function*). Click on the function button **Delete**. A dialog box will warn you that you are about to delete a function.

Click on the command button **Yes** to physically erase the function from the database or **No** if you do not want to erase. If you proceed with the delete operation, the function is removed from the database and the editor will have no function under edition.

Notes:

- If you simply want to remove the function from the editor, use the closure instead. (See *Close a function*).
- Usually, the entity manager is responsible for the edition of labels and descriptions, and of deleting entities. In this case, these operations are done locally to remain consistent with the fact that function labels cannot be edited in the entity manager. (You can do it, but you should not...)

Compile a function

You have access to a syntax analysis utility to make sure that your function has no syntax error.

To make a syntax analysis of your function, click on the function button **Compile** at the bottom of the editor. After the analysis, the software will display an error message if there is any syntax problem, or a message telling you that the syntax seems correct.

Example:

```
function scalar test ()
 test = "abc"
end function
```

The **test** function above should return a **scalar** but intentionally we make it return a **string**. The syntax analysis finds the problem and informs you :

```
Syntax error
ERR_CALCUL_LIGNE: 2
test = "abc"?
```

This type of error is detected. If the syntax analysis utility finds nothing wrong with your function, you will have a message mentioning it.

Cut selected code

You have access to the basic functions of a text editor, i. e. cut, copy, paste and cancel.

To cut code lines, first select what you want to cut, and press **Ctrl+X** or **Maj+Suppr** on the keyboard to cut the code selected. You can paste it elsewhere.

See also:

- Copy selected code
- Paste code
- Cancel an operation

Copy selected code

You have access to the basic functions of a text editor, i. e. cut, copy, paste and cancel.

To copy code lines, first select what you want to copy, and press **Ctrl+C** ou **Ctrl+Insert** on the keyboard to copy the code selected. You can paste it elsewhere.

See also:

- Copy selected code
- Paste code
- Cancel an operation

Paste code

You have access to the basic functions of a text editor, i. e. cut, copy, paste and cancel.

To paste code lines, you must have previously copied or cut code lines. Position the cursor in the text at the location where you want to paste the code. Press **Ctrl+V** or **Maj+Insert** on the keyboard to paste the code.

See also:

- Copy selected code
- Paste code
- Cancel an operation

Cancel an operation

You have access to the basic functions of a text editor, i. e. cut, copy, paste and cancel.

To cancel the last operation, press **Ctrl+Z** on the keyboard. You have only one level of cancel.

Projection tool

The projection tool provides the possibility to project a field generated on a given mesh toward another mesh, thus creating a new field of the same type.

The domain of the destination mesh, i.e. the surface covered by it must be compatible with the source mesh. The destination mesh must be inside the domain of the source mesh.

If the destination mesh cannot obtain values for all its nodes, the projection tool will not be able to generate adequately a new field. It will inform you of the situation.

To project a field toward another mesh, follow these steps :

1 - In the **Tools** menu, select the item **Projection...**. A **Mesh projection** dialog box appears.

2 - The box contains two large zones, the source field and the destination field.

3 - Select the source mesh and field in the first group (See *Specify the source field*.)

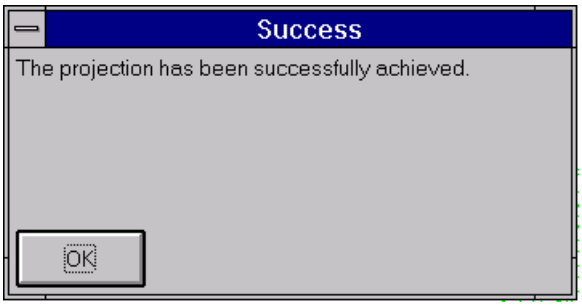
4 - Select the destination mesh and specify the label of the destination field. (See *Specify a destination field*).

5 - Click on **OK** to launch the projection or **Cancel** to cancel the projection.

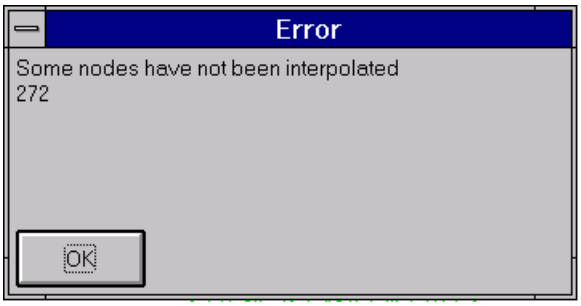
6 - The software will inform you if the projection has been successful or not.

Notes :

- If the projection has been successful, the software tells you so.



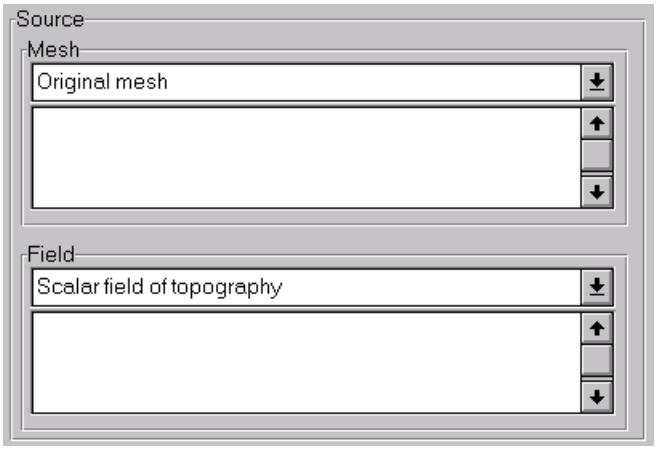
- If it failed, the software indicates what did not work, and save the new field nevertheless.



- The type of the source field is preserved during the projection. If the source field is a Scalar field - topography, the new field will be of the same type.
- All the types of fields can be projected.

Specify the source field

The specification of a source field determines the data that will be projected. The specification page has two sections : the source mesh and the source field.



In the drop-down list of **meshes**, select the mesh on which the field to be projected is generated. The list contains all the meshes present in your project.

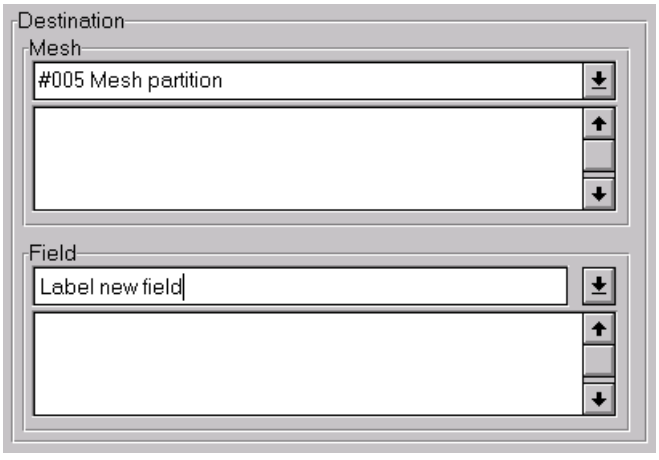
In the drop-down list of **fields**, select the field that you wish to project. The list contains all the fields available on the mesh selected.

Notes :

- All the types of fields are available : scalar, vector, and others.

Specify a destination field

The specification of a destination field determines on which mesh the source field will be projected. The specification page has two sections : the destination mesh, and the identification of the new field.



In the drop-down list of **meshes,** select the mesh on which you want to project the source field. The list contains all the meshes available in your project.

Enter the identification of the new field in the appropriate section. You can draw inspiration from the existing labels for this type of data. This identification will be given to the new entity created in the database.

Warning :

- The destination mesh must be compatible with the source mesh in terms of domain covered. The destination mesh must be inside the source mesh, otherwise the projected field will contain values that were not interpolated. These values will take the value of a reference value.

Profile mode

The profile mode specifies transects in the form of profile partition. The profile partition is a specialized polyline carrying information at the level of the vertexes of the polyline.

Prequisite : Before generating a profile, the following conditions must be satisfied :

- a project must be loaded (See the documentation on project managementAID_PRJ_CHARGE@gestproj.hlp)
- at least one display window must exist (See the documentation on display controlAID_MNU_FEN_CREE@ctlaffic.hlp)

To accede to this mode, in the **Tools** menu, select the item **Profile [...]**.

The functions in the profile mode are divided into two categories :

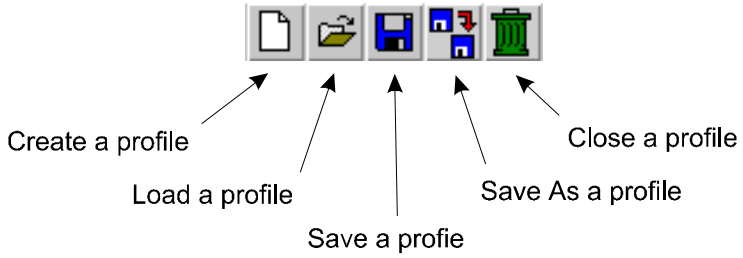
1 - Profile management

2 - Profile edition.

Profile management

A series of function buttons appears at the bottom of the screen when you enter the profile mode.

Figure 17 Profile management function button toolbar



In the profile mode, you can create, load, save, save as, and close a profile. A profile comprises a series of segments forming a transect.

The following sections address the profile mode functions :

- Create a profile
- Load a profile
- Save a profile
- Save As... a profile
- Close a profile.

Create a profile

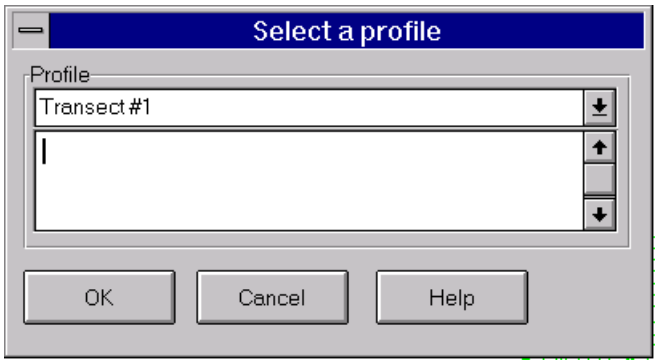
To create a profile, click on the function button in the profile management function toolbar.

If a profile was already loaded, you will be asked to save it and it will be closed and replaced by the new profile.

To edit the profile, see *Profile edition*.

Load a profile

To load a profile, click on the function button in the profile management function toolbar. A profile selection dialog box appears.



In the drop-down list, select the profile of your choice. Click on **OK** to load this profile or on **Cancel** if you do not want to load it.

If a profile was already loaded, you will be asked to save it and it will be closed and replaced by the new profile that you are loading.

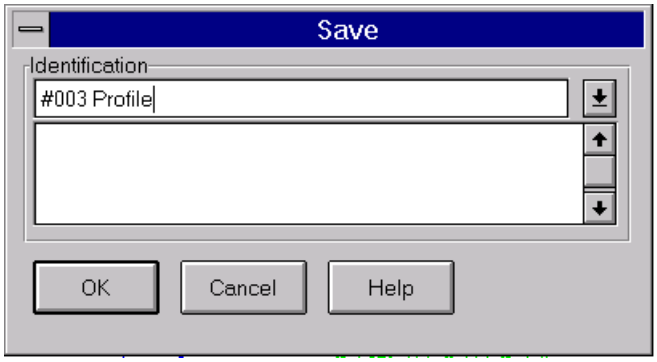
To edit the profile, see *Profile edition*

Save a profile

To save a profile, click on the function button in the profile management function toolbar.

If the profile has already been saved, the profile is saved immediately. An hourglass appears, and when it disappears, the profile has been saved.

If the profile has never been saved, you will be asked to provide a label and a description for the new profile.



Enter a label in the upper part of the group **Identification**. You can draw inspiration from existing labels for this type of data. However, you must supply a unique label.

Enter a description in the lower part of the group **Identification**.

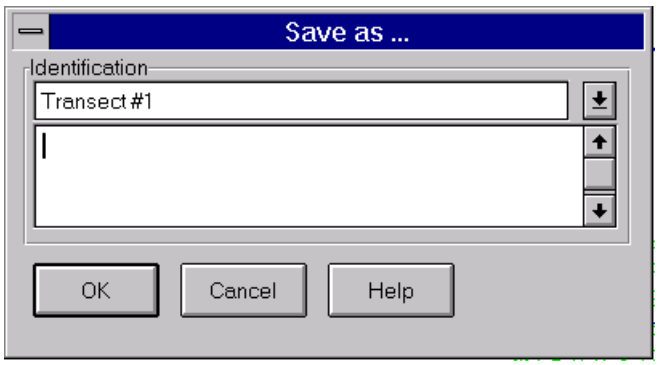
Click on **OK** to confirm the operation or on **Cancel** if you do not want to save the profile.

Save As... a profile

Saving a profile under a different name (Save as...) creates a complete copy of the current profile. To save a profile under a different name, click on the function button in the profile management function toolbar.

If the profile has never been saved, the procedure will be the same as with a save operation. (See *Save a profile*.)

If the profile has previously been saved, the Save As... dialog box appears.



You must provide a unique label for this type of data in the upper part of the group **Identification**. The label displayed is that of the profile to be saved as..., and it cannot be used as such.

You may draw inspiration from it or from any other to write a new unique label.

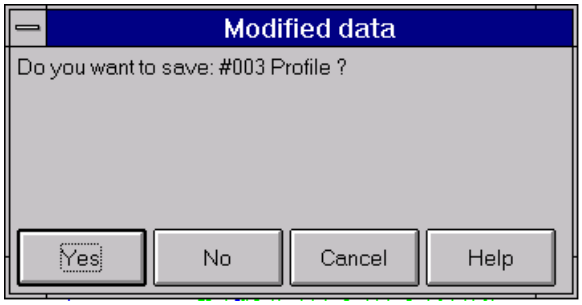
Enter a description in the lower part of the group **Identification**.

Click on **OK** to confirm the save as... operation or on **Cancel** to cancel the operation. The profile saved as... becomes the current profile.

Close a profile

To remove a profile from the display, you must absolutely follow this procedure. Click on the function button in the profile management function toolbar.

If the entity has been modified, a pop-up window appears and inform you of the situation.



Click on **Yes** to save the modifications, **No** if you do not want to save the modifications and **Cancel** to cancel the closure.

Notes :

- This operation does not destroy the profile, it simply removes it from the display. Since a profile is not a static entity because we can interact with its elements, the entity cannot be closed in the **Management of the display list** window. (See the documentation on Display control AID_LISTE_FENETRES@ctlaffic.hlp.)
- To delete a profile from the database, you must go to the **Entity manager** in **Project management**. (See the documentation on project management AID_GESTIONNAIRE_PROJET@gestproj.hlp)

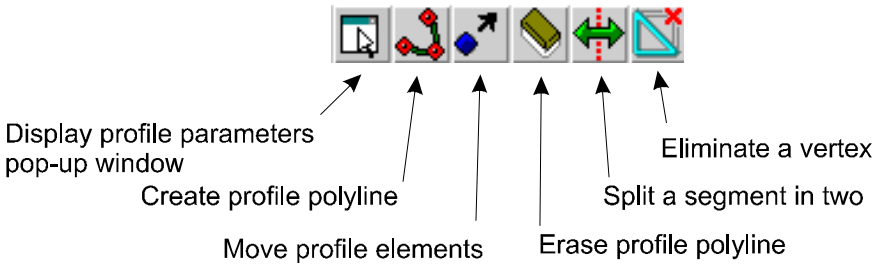
Profile edition

A series of function buttons are available at the bottom of the screen for the edition of profiles.

Prerequisite :

- To edit a profile, you must have created or loaded a profile. (See *Profile management*.)

Figure 18 Profile edition function buttons toolbar



Edition is done on a profile already loaded. The following operations are possible on a profile :

- Display the parameters window
- Create the polyline of the profile
- Move the elements of the profile
- Erase the polyline of the profile
- Split a segment in two
- Eliminate a vertex from a polyline.

Display the parameters window

To display the profile parameters window, click on the function button in the profile edition toolbar. A pop-up window appears..

This window is designed to work in two modes :

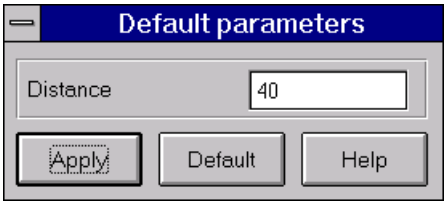
1 - Default parameters

2 - Parameters of a vertex.

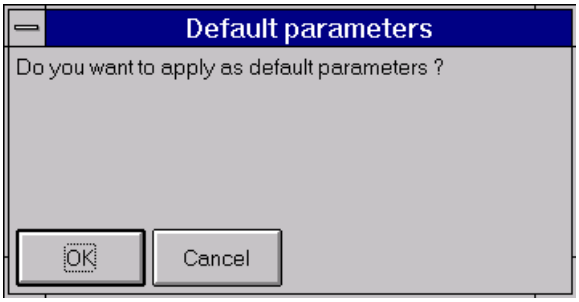
Default parameters

The profile parameters pop-up window is in the default parameters mode when there are no elements of the profile selected.

If neither vertexes nor edges have been selected on the profile, the window looks like this.



At this point, you can specify which default value to assign to a new vertex. Click on the command button **Apply** to assign the default value. The confirmation box appears.



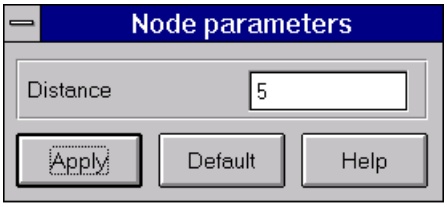
Click on **OK** to apply the default value or on **Cancel** if you do not want to apply it.

The next vertexes created will have this default value.

Parameters of a vertex

The profile parameters pop-up window is in the parameters of a vertex mode when a vertex of the profile has been selected.

If a vertex or an edge has been selected on the profile, the window looks like this.



You can modify the value and click on **Apply** to apply the value on the selection.

You can also click on **Default** to obtain the default value and then click on **Apply** to apply the default value on the selection.

Notes :

- The value carried by the vertex corresponds to the desired spacing between the values evaluated during the drawing of the profile in the static display mode. (See the documentation on profile display AID_FEN_PROFIL_EN_LONG@visudonn.hlp.)
- The unit of the value is the abstract unit of the project in the case of a non georeferenced project, and the meter otherwise.

- If you select an edge and the values carried at the two vertexes differ, the interface will display dash lines, However, you can enter a value and apply it to both vertexes.

Create the polyline of the profile

To create the polyline of the profile :

1 - Click on the function button in the profile edition toolbar. You then enter a special mode of creation of the polyline.

2 - Click with the mouse in the display window to create a first point. This first point has a special marker indicating the starting point of the transect (a small empty circle with a star inside).

3 - Then click with the mouse in the display window to create the following points, At each mouse click, a vertex and a segment linking this vertex to the preceding vertex are created.

4 - to end the creation of the polyline, click again on the function button in the profile edition toolbar. This ends the entry. You can also end the entry by double-licking on the last point entered.

5 - The default parameters are assigned to all the vertexes.

Notes :

- When you are in the polyline creation mode, all the other functions are deactivated. However you can make zooms, re-center, etc. allowing you to complete the entry of the vertexes.
- The other edition functions will allow you to make corrections to your entry.
- If you re-enter the polyline creation mode while there is already a polyline created, the entry can proceed from the last point entered.

Move the elements of the profile

You can move a vertex or an edge of your polyline at will.

To move an element of a polyline :

1 - Click on the function button of the profile edition toolbar. You enter a special edition mode.

2 - Select a vertex or a crest and use the drag and drop technique to move the element to the desired position.

3 - Release the mouse button to confirm the displacement.

4 - Repeat the operation at will.

5 - To exit this mode, click again on the function button in the profile edition toolbar.

Note :

- When you are in the displacement mode, the other functions are deactivated. However you can make zooms, re-center, etc. allowing you to complete your displacements.

Erase the polyline of the profile

You can erase the entire polyline by clicking the function button in the profile edition toolbar. The polyline is immediately erased.

Notes :

- You will not be asked to confirm.
- You cannot cancel the erase command.

Split a segment in two

You can split a segment in two sections by adding a new vertex in the center of the edge :

1 - Select an edge on your polyline.

2 - Click on the function button in the profile edition toolbar

3 - A new vertex is added at the center of the edge.

Notes :

- You can remove this vertex, see *Eliminate a vertex from a polyline*.
- This new vertex can be moved like any other vertex.

Eliminate a vertex from a polyline

You can eliminate a vertex from a polyline as long as it is neither at the beginning nor at the end of the polyline. To eliminate a vertex :

1 - Select a vertex of the polyline (interior vertex)

2 - Click on the function button to eliminate the vertex.

3 - The vertex is removed and the polyline connection between remaining vertexes is reestablished.

Chapter 9 The data model

The software provides a highly performing work environment in which you can conduct hydrodynamic and/or environmental studies based on a solution using the finite elements method.

The database releases you from the management of the files needed for hydrodynamic studies and support you in your work, since the nature and complexity of the data involved can rapidly become unmanageable. In addition, the database preserves the integrity of the data.

The software creates a database grouping the data in the form of projects containing interrelated entities. In this context, you never have to open, write, create or delete data files ; you interact with projects containing entities which represent the data. These entities maintain links between them, which may be compulsory or optional links as defined by the data model.

The database makes sure that the data are always coherent, complete and valid in compliance with the data model. Thus you benefit of a support allowing you to conduct analysis and modeling tasks in the most harmonious way possible.

The database supports the concept of the type of data with which you will have to interact. All through a study using the software, you will have to import and to export information. You will create a wide range of data to build the numerical terrain model ; you will construct simulations and recover the results.

This section defines each type of data, the position they occupy in the data flow, and the links that they must maintain with other data.

Data generated by **field surveys**:

- Field - percentage of substrates
- Scalar field - topography terrain
- Substrate classes
- Coordinate system
- Data points
- Topographic terrain
- Substrates terrain

Data generated by the **construction of the numerical terrain model**:

- Field - ice
- Scalar field - topography
- Scalar field - Manning friction
- Scalar field - substrate friction
- Vector field - wind

- Boundary conditions
- Mesh
- Partition mesh
- Boundary conditions partition
- Manning friction partition
- Ice partition
- Mesh partition
- Substrates partition
- Topography partition
- Wind partition

Data generated by **simulation results** :

- Scalar field - local mass balance
- Scalar field - relative mass balance
- Scalar field - error of water level
- Scalar field - error of specific discharge in X
- Scalar field - error of specific discharge in Y
- Scalar field - surface level
- Scalar field - Froude number
- Scalar field - depth
- Scalar field - residuals of specific discharge in X
- Scalar field - residuals of specific discharge in Y
- Scalar field - residuals of water level
- Scalar field - shear stress velocity
- Scalar field - numerical viscosity
- Scalar field - turbulent viscosity
- Scalar field - velocity modulus
- Vector field - velocity
- Hydrodynamic simulation

Other data :

- Reference event
- System function

- User function

Field

The notion de *field* is central in a software using the finite elements method. The *field* is in fact a vector of values associated with each node of the *Mesh* to which the field is linked. The field always has exactly the same number of elements as there are nodes in the associated mesh.

There are different categories of field in our software.

1. The *Scalar* which contains a single scalar value per node.
2. The *Vector field* which contains three scalar values per node, i. e. the components x, y and z.
3. The *Field - N values* which may contain N scalar values per node.

In the software, several types of field in one of the three categories above have been defined to characterize precisely the data..

Scalar field

The *scalar field* is the common form of *Field* It is a vector of scalar values associated with each node of the related mesh. The field must contain the same number of elements as there are nodes in the associated *Mesh*.

The software use several scalar fields in different contexts : source of data, results, post-treatment... Several specific types have been defined to characterize the origin and the nature of the data available.

Here is the list of the scalar fields :

- Scalar field - topography
- Scalar field - surface level
- Scalar field - depth
- Scalar field - velocity modulus
- Scalar field - Froude number
- Scalar field - shear stress velocity
- Scalar field - turbulent viscosity
- Scalar field - numerical viscosity
- Scalar field - local mass balance
- Scalar field - relative mass balance
- Scalar field - residuals of specific discharge in X
- Scalar field - residuals of specific discharge in Y
- Scalar field - residuals of water level

- Scalar field - error of water level
- Scalar field - error of specific discharge in X
- Scalar field - error of specific discharge in Y
- Scalar field - Manning friction
- Scalar field - substrate friction
- Scalar field - topography terrain

Scalar field - topography

The *scalar field - topography* designates the topography transferred on a *Mesh* used for the *hydrodynamic simulation*. This field defines the topography of the study domain ; this data is produced using the topography mode of MODELEUR in which information on topography from several heterogeneous sources can be assemble using a *Topography partition*.

This data is **compulsory** to execute a hydrodynamic simulation.

Position in the data flow

The scalar field - topography is generated during the conception of the numerical terrain model after the *Mesh* for the hydrodynamic simulation have been created, and after all the information needed on the topography, covering the entire study domain, has been gathered.

The main source of topography data is the *Topographic terrain* which has been generated during the importation of the topography data. It is possible to modify the original topography by modifying locally a part of the river bed, using a *Function* which can be interpreted by the programmable calculator.

The hydrodynamic simulation uses this data to establish the morphology of the domain to be simulated.

Direct links

- The *scalar field - topography* is always linked to the *Mesh*.
- The *scalar field - topography* remains linked to the *Topography partition* from which it was generated.

Indirect links

- The *Hydrodynamic simulation* always keeps a link with a *scalar field - topography* since this data is **compulsory**.

Scalar field - surface level

The *scalar field - level surface* designates a result of the *Hydrodynamic simulation*. The level of the free surface is one of the important results of the simulation. Two types of data are related to it : the *Scalar field - depth* and the *Scalar field - topography* (See Figure 19). The latter is an input data for the *simulation* and the first is calculated as a function of the other two in the following relation :

```
Depth = Surface level - Topography.
```

Position in the data flow

The *scalar field - surface level* is generated by the simulation and is therefore considered a calculation result. This result can be used later to conduct studies using the results of river hydrodynamic simulations.

Direct links

- The *scalar field - surface level* is always linked to a *Mesh*.

Indirect links

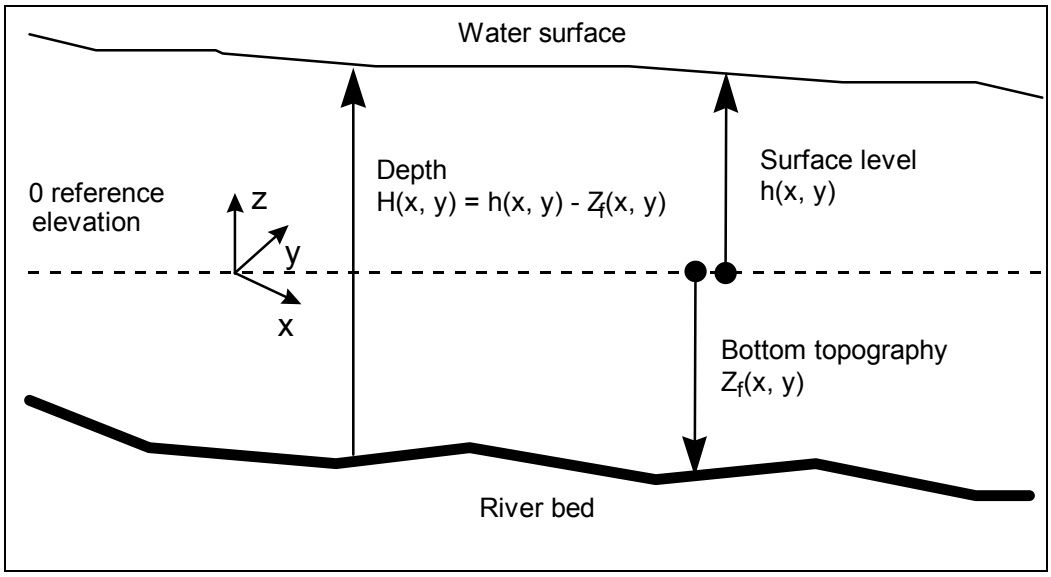
- Eventually, if the results of a *simulation* are retained, the *scalar field - surface level* will be referred by a *Reference event*.

Scalar field - depth

The *scalar field - depth* designates a result generated by the hydrodynamic simulation. The *depth* is an important result of the simulation and is calculated using the *Scalar field - topography* and the *Scalar field - surface level* through the following relation:

```
Depth = Surface level - Topography.
```

Figure 19 Relation Depth/Surface level



Position in the data flow

The *scalar field - depth* is generated by the simulation and is therefore considered a calculation result. This result can be used later to conduct studies using the results of river hydrodynamic simulations.

Direct links

- The *scalar field - depth* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained, the *scalar field - depth* will be referred by a *Reference event*.

Scalar field - velocity modulus

The *scalar field - velocity modulus* is a calculation result of the *Hydrodynamic simulation*. This field is one of the important results of the hydrodynamic simulation used in a fair number of analyses.

Position in the data flow

The *scalar field - velocity modulus* is a calculation result of the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- A *scalar field - velocity modulus* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained, this scalar field will be referred by a *Reference event*

Scalar field - Froude number

The *scalar field - Froude number* is a calculation result of the *Hydrodynamic simulation*. It represents the ratio between the forces of inertia and the forces of gravity. In hydraulics, the Froude number classifies the flows in three categories:

1. froude < 1 : sub-critical flow
2. froude = 1 : critical flow
3. froude > 1 : rapid flow.

A rapid flow simply means that the constraints downstream have no influence on the flow upstream because the slope is too steep.

Position in the data flow

The *scalar field - Froude number* is a result of the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- A *scalar field - Froude number* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained, the *scalar field - depth* will be referred by a *Reference event*.

Scalar field - shear stress velocity

The *scalar field - shear stress velocity* is a result of the *Hydrodynamic simulation*.

Position in the data flow

The *scalar field - shear stress velocity* is a result of the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - shear stress velocity* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*.

Scalar field - turbulent viscosity

The *scalar field - turbulent viscosity* is a result produced by the *Hydrodynamic simulation*.

Position in the data flow

The *scalar field - turbulent viscosity* is a result of the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - turbulent viscosity* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event.*

Scalar field - numerical viscosity

The *scalar field - numerical viscosity* is a result produced by the *Hydrodynamic simulation.*

Position in the data flow

The *scalar field - numerical viscosity* is a result produced by the *Hydrodynamic simulation.* This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - numerical viscosity* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*.

Scalar field - local mass balance

The *scalar field - local mass balance* is a result produced by the *Hydrodynamic simulation*

Position in the data flow

The *scalar field - local mass balance* is a result produced by the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - local mass balance* is always linked to a *Mesh*..

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*.

Scalar field - relative mass balance

The *scalar field - relative mass balance* is a result produced by the *Hydrodynamic simulation*.

Position in the data flow

The *scalar field - relative mass balance* is a result produced by the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - relative mass balance* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*.

Scalar field - residuals of specific discharge in X

The *scalar field - residuals of specific discharge in X* is a result produced by the *Hydrodynamic simulation.*

Position in the data flow

The *scalar field - residuals of specific discharge in X* is a result produced by the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - residuals of specific discharge in X* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*

Scalar field - residuals of specific discharge in Y

The *scalar field - residuals of specific discharge in Y* is a result produced by the *Hydrodynamic simulation.*

Position in the data flow

The *scalar field - residuals of specific discharge in Y* is a result produced by the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - residuals of specific discharge in Y* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*

Scalar field - residuals of water level

The *scalar field - residual of water level* is a result produced by the *Hydrodynamic simulation*

Position in the data flow

The *scalar field - residual of water level* is a result produced by the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - residual of water level* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*.

Scalar field - error of water level

The *scalar field - error of water level* is a result produced by the *Hydrodynamic simulation.*

Position in the data flow

The *scalar field - error of water level* is a result produced by the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - error of water level* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*.

Scalar field - error of specific discharge in X

The *scalar field - error of specific discharge in X* is a result produced by the *Hydrodynamic simulation*.

Position in the data flow

The *scalar field - error of specific discharge in X* is a result produced by the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - error of specific discharge in X* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*.

Scalar field - error of specific discharge in Y

The *scalar field - error of specific discharge in Y* is a result produced by the *Hydrodynamic simulation*.

Position in the data flow

The *scalar field - error of specific discharge in Y* is a result produced by the *Hydrodynamic simulation*. This type of data appears when the simulation results are recovered, and this field is used when interpreting the results.

Direct links

- The *scalar field - error of specific discharge in Y* is always linked to a *Mesh*.

Indirect links

- Eventually, if the results of a simulation are retained this scalar field will be referred by a *Reference event*.

Scalar field - Manning friction

The *scalar field - Manning friction* contains the friction values directly specified with a *Manning friction partition*. The Manning friction partition specifies friction by sub-domains by directly giving a Manning coefficient. This allows to adjust friction values at specific locations to obtain a more precise configuration of the numerical terrain model., if needed. This operation should be executed last.

The *scalar field - Manning friction* is generated on a given *Mesh* and a Manning friction value is given to each node of the mesh.

Position in the data flow

The *scalar field - Manning friction* is a data created during the construction of the numerical terrain model. After building a *Manning friction partition* and specifying friction values over the entire study domain, this data can be generated on a given mesh.

It is at that time that the *scalar field - Manning friction* appears in the data flow. (See Figure 29 Data flow for the **Manning**). It will eventually be used in the *Hydrodynamic simulation* in the section of friction limiting water flow.

Direct links

- The *scalar field - Manning friction* is always linked to a *Mesh*.
- The *scalar field - Manning friction* always keeps a link with the *Manning friction partition* from which it was generated.

Indirect links

- If this friction scalar field has been used in a *Hydrodynamic simulation*, the simulation may be linked to it.

Scalar field - substrate friction

The *scalar field - substrate friction* contains the friction values related to the substrates specified with a *Substrates partition*. The *substrates partition* characterizes the composition of the river bed by keeping the proportions of every *substrate classes* by sub-domain.

The *scalar field - substrate friction* is generated on a given *Mesh* and a substrate friction value is given to each node of the *mesh*.

Position in the data flow

The *scalar field - substrate friction* is a data created during the construction of the numerical terrain model. After having specified the composition of the river bed in the substrates mode with a *Substrates partition*, friction values can be generated on a mesh as a function of the river bed composition.

The *scalar field - substrate friction* will then appear in the data flow. This field will be used in the *Hydrodynamic simulation* in the section of friction limiting water flow (See Figure 21 data flow for **substrates**).

Direct links

- The *scalar field - substrate friction* always refers to the *Mesh* to which it is linked.
- The *scalar field - substrate friction* also keeps the link with the *Substrates partition* from which it was generated.

Indirect links

- If this friction scalar field has been used in a *Hydrodynamic simulation*, the simulation may be linked to it.

Scalar field - topography terrain

The *scalar field - topography terrain* designates the topographic values from field surveys. It is always associated with *Data points* (or to a *Mesh*) with a corresponding

number of nodes. Thus the *scalar field - topography terrain* provides a true value for each node of the related data points.

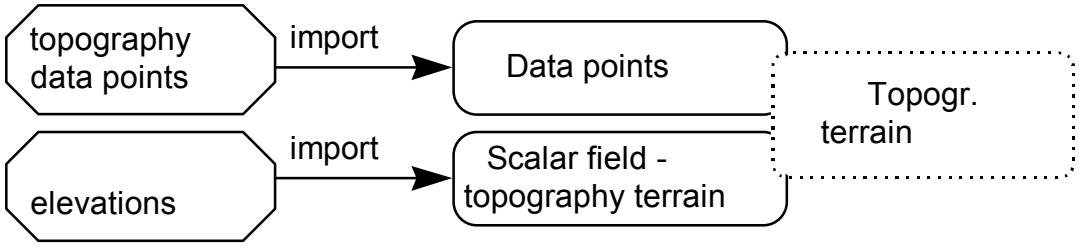
The *scalar field - topography terrain* is automatically linked to a *Topographic terrain* which aggregates the *data points* and the *scalar field - topography terrain*.

Special attention should be given to make sure that the topographic data are normalized under the same 0 reference level for the entire study domain.

Position in the data flow

The *scalar field - topography terrain* is a terrain data created when importing topographic data. This terrain data associated to *data points* or to a *mesh* will generate a *Topographic terrain* when imported.

Figure 20 Topography terrain data flow



The *scalar field - topography terrain* will be used in the construction of the numerical terrain model. In the topography mode, a *Topography partition* will refer to *topographic fields*.

Direct links

- The *scalar field - topography terrain* is always linked to *Data points* or to a *Mesh*.

Indirect links

- A *Topographic terrain* always refers to the *scalar field - topography terrain* .

Vector field

The *vector field* designates the category of fields which comprise three scalar components x, y and z. In two dimension modeling, The scalar component x and y are used.

The following vector fields are used in the software :

- *Vector field - velocity*
- *Vector field - wind*.

Vector field - velocity

The *vector field - velocity* designates a velocity field resulting from the *Hydrodynamic simulation*. This field is one of the important results produced by the simulation ; with the *vector field - velocity* and the *Scalar field - depth,* detailed analyses can be conducted.

A *vector field - velocity* divides the velocity in components x and y.

Position in the data flow

The *vector field - velocity* is a result of the *Hydrodynamic simulation*. Thus, a velocity field is generated on the *Mesh* used for this simulation.

This data can be used as a final product as such, or be used for environmental simulations using river hydrodynamics.

Direct links

- The *vector field - velocity* is always linked to a *Mesh*.

Indirect links

- If the results of the simulation are retained, this field will be referred by a *Reference event*.

Vector field - wind

The *vector field - wind* designates a data generated during the construction of the numerical terrain model. Wind characteristics are relevant in modeling when the winds can affect the river flow.

Position in the data flow

In the wind mode, a *Wind partition* is built with which wind direction and strength can be specified for the study domain. The *vector field - wind* can be generated on a designated *Mesh*.

This data can then be used in a *Hydrodynamic simulation* allowing to specify the constraints to river flow induced by the dominant winds.

Direct links

- The *vector field - wind* is always linked to a *Mesh*.
- The *vector field - wind* also keeps a link with the *Wind partition* from which it was generated.

Indirect links

- If the *vector field - wind* is used in a *Hydrodynamic simulation*, the simulation will remain linked to the *vector field - wind*.

Field - N values vector

This generic field defines the other possibilities in terms of field, i.e. nodal values other than scalar or vector values. The following fields are in this category :

- *Field - ice*
- *Field - percentage of substrates*

Field - ice

The *field - ice* designates a data generated during the construction of the numerical terrain model. The *field - ice* contains the specification of ice cover for each node of the designated *Mesh*.

The specification of ice contains two pieces of information interesting for the *Hydrodynamic simulation*:

- The ice thickness
- Ice friction related to the type of ice.

Position in the data flow

The *field - ice* appears during the construction of the numerical terrain model. In the ice mode, an *Ice partition* is created, with which the characteristics of the ice, thickness and friction, can be specified by sub-domain.

Once the study domain has been correctly specified, the specifications can be transferred on a designated *Mesh*. The *field - ice* will then be used by a *Hydrodynamic simulation* which will consider the ice features to adjust the water flow.

Direct links

- The *field - ice* is always linked to a *Mesh*.
- The *field - ice* also keeps a link with the *Ice partition* from which it was generated.

Indirect links

- If this field is used in a *Hydrodynamic simulation*, the simulation will remain linked to the *field - ice*.

Field - percentage of substrates

The *field - percentage of substrates* designates the field of percentages of substrates classified by category, a field generated from terrain data. It is always associated to *Data points* (or a *Mesh*) with a corresponding number of nodes. It is created to be able to use terrain data specifying the nature of substrates at survey points.

The *field - percentage of substrates* is automatically associated to a *Substrates terrain* which aggregates the *data points* and this *field - percentage of substrates*.

The field will use *Substrate classes* to distribute the proportions of various grain sizes found at each survey point into 7 categories of substrates.

Position in the data flow

The *field - percentage of substrates* is part of the data collected on the terrain. This type of data appears during the importation of terrain data on substrates.

Following the importation of this type of data and the automatic creation of a *Substrates terrain*, this data will be used for the construction of the numerical terrain model.

Thus, in the substrate mode, a *Substrates partition* is created based on the same *Substrate classes* and the *fields - substrates* are used to specify the nature of the substrates over the entire study domain. (See Figure 21).

Direct links

- The *field - percentage of substrates* is always linked to a *Mesh* or to *Data points*.
- The *field - percentage of substrates* must absolutely be linked to the designated *Substrate classes* to know the distribution of grain sizes.

Indirect links

- The *Substrates terrain* always refers to a *field - percentage of substrates*

Substrate classes

The *substrate classes* defines the classification of the river bed substrates. The classification includes 7 categories of grain sizes based on 7 diameter ranges.

This classification must be part of the terrain data and should be decided prior to field work. The substrate classes can be imported in the data base using the import mechanism.

The *substrate classes* defines the ranges of diameters which will be used to specify the substrate during the construction of the numerical terrain model.

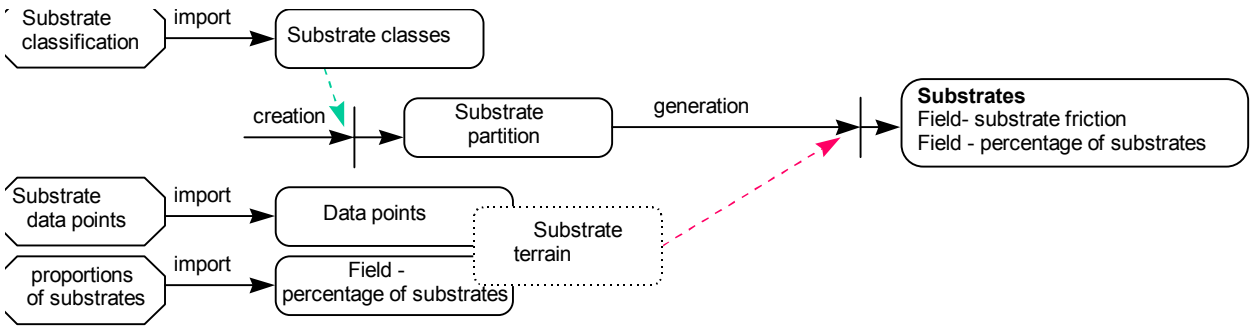
Position in the data flow

The *substrate classes* must be specified before field work is conducted. It is essential to have a classification *before* the terrain is characterized in terms of substrate.

The *substrate classes* must also be integrated to the project *before* the importation of terrain data on substrates.

When creating a *Substrates partition*, a *substrate classes* with which you can work, must necessarily be provided. It is of primary importance to remain coherent overt the entire domain with substrate classifications.

Figure 21 data flow for substrates



Direct links

The *substrate classes* are never linked to any other entity.

Indirect links

A *Substrates partition* is always linked to the *substrate classes*.

A *Substrates terrain* is always linked to the *substrate classes*.

Boundary conditions

The specification of *boundary conditions* is a compulsory operation to execute a numerical *Hydrodynamic* simulation. For the numerical simulation to lead to a solution (convergence), it is essential to specify the physical values on the contour and inside of the simulation domain.

In the case of a hydrodynamic simulation, the contour is determined by the shores and the open boundaries, if any. Usually, an impermeability condition and a sliding condition are imposed on the shores. (This operation is conducted *by default* in MODELEUR, i.e. nothing has to be specified, for greater utilization flexibility).

As for open boundaries, it is generally required to impose the water level on the downstream open boundary to regulate the level of the water body. On the upstream boundary, you have the choice to impose either a water level or the inflow.

The *boundary conditions* are a result generated by MODELEUR in the mode of boundary conditions specification. This result is generated on the basis of constraints specified in the *Boundary conditions partition* which represents the skin or the contour of the study domain.

This result is an input data for the simulator HYDROSIM.

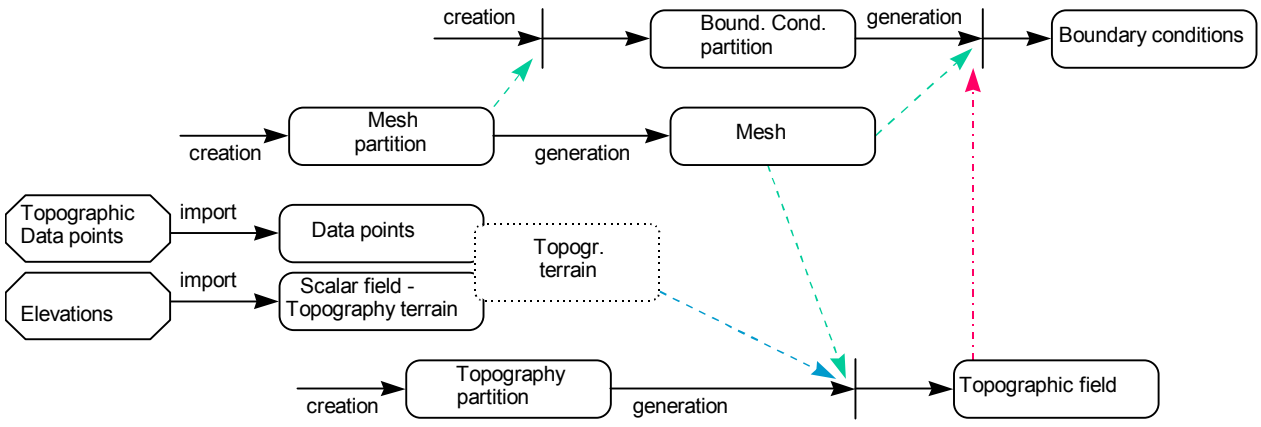
For a more complete description of the physics of a two-dimensional hydrodynamic simulation, see HYDROSIM User's Guide.

Position in the data flow

A data of the type *boundary conditions* is generated from a *Boundary conditions partition* after the user has specified constraints at the boundaries of the domain. First, the user must have generated a *Partition* from a *Mesh partition*. It may be necessary to dispose of the topography to be used with the mesh, if the boundary conditions constraints are specified in terms of discharge volume. You must then link to a *Scalar field - topography* based on the same mesh.

These boundary conditions can be put in the specifications of a simulation to launch a *Hydrodynamic* simulation with HYDROSIM.

Figure 22 Boundary conditions data flow



Direct links

- The *boundary conditions* are linked to a *Partition mesh*.

- The *boundary conditions* **may** be linked to a *Scalar field - topography*
- The *boundary conditions* are linked to a *Boundary conditions partition*.

Indirect links

- A *Hydrodynamic* simulation may refer to *boundary conditions*.

Reference event

The *reference event* designates the end product exported by MODELEUR to be used by other environmental analysis software. The *reference event* will soon be defined...

Function

The *function* is a data widely used by the calculator module ; its structure is that of a program written in the language supported by the calculator.

The user can write various programs allowing him to process data from the database.

There are two categories of functions:

- System function
- User function.

System function

The *system function* designates the functions offered by the system, to which the user has access, but that he cannot modify.

User function

The *user function* designates the functions written by the user which can be later modified.

Partition mesh

The *partition mesh* is a standard *Mesh* with the characteristic of having been generated from a *Mesh partition*, i.e. generated by MODELEUR.

A link is maintained with the *mesh partition* from which it was generated. The *partition mesh* is an input data for the *Hydrodynamic simulation*.

Mesh

The *mesh* is the basic entity allowing the implementation of a system of spatial representation based on finite elements. By definition, a mesh contains a series of *nodes* defining points located in space and a series of *elements* establishing connections between *nodes*, forming the complete mesh.

A *mesh* is always linked to a *Coordinate system* to provide for accurate localization in the space of representation.

Position in the data flow

The *mesh* occupies a central position in the hydrodynamic modeling based on *finite elements*. A well constructed mesh can be imported from outside of the software by the import procedure, and meshes can be created with MODELEUR in the mesh mode using a *Mesh partition*.

A *mesh* may comprise different types of elements, but the only element admissible for the *Hydrodynamic simulation* is the *T6L element,* since this element is the most adapted to the simulation executed by HYDROSIM.

Direct links

- A *mesh* always refers to a *Coordinate system* to specify the spatial representation.

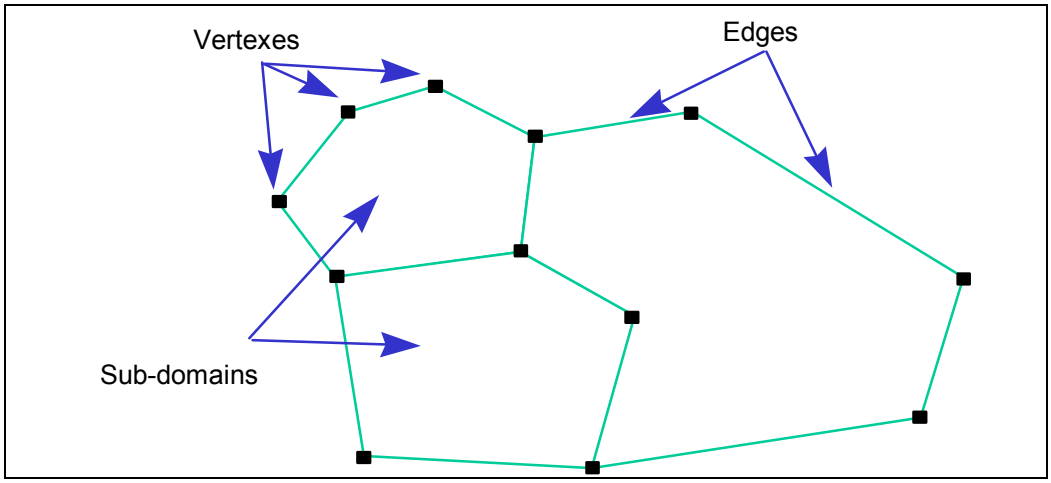
Indirect links

- A *mesh* can be referred by several types of data. In fact, any *Field* can refer to a *mesh*.

Partition

The *partition* is a spatial tool of specification composed of *vertexes*, *edges* and *sub-domains*. The *partition* includes a certain number of constraints to remain in a coherent state. For example, an *edge* cannot cross another one, a *sub-domain* must be a closed surface and never be inside another *sub-domain*. The *partition* has the use of several functions allowing to construct and modify its geometry.

Figure 23 The structure of a partition



The *partition* is a generic tool designed to supply information on the *vertexes*, *crests* and *sub-domains*. There is a certain number of specific *partitions* in the software, and each one carries specific information and is designed to generate specific data. The strength of the partition is that it allows the *independence* between terrain data and the data actually used in the hydrodynamic simulation.

Following is an exhaustive list of the partitions available:

- The *Mesh partition*

- The *Topography partition*
- The *Boundary conditions partition*
- The *Substrates partition*
- The *Wind partition*
- The *Ice partition*
- The *Manning friction partition*

Mesh partition

The *mesh partition* is a component presenting all the features of the generic *Partition* with, in addition, a data support allowing the specification of the criteria for *Mesh* generation.

The user can build a *mesh partition* by creating the sub-domains needed to cover the study domain. The *vertexes* of the *Partition* will carry the information on the spacing wanted (in meters) between the *nodes* of the *Mesh*. The result is an adapted mesh on which the local density of the *nodes* can be controlled.

For the moment, the *mesh partition* generates *meshes* with T3, T6 and T6L elements. The partition generates *Partition* which become input data for the simulator HYDROSIM, and on which is transferred all the information required for the simulation.

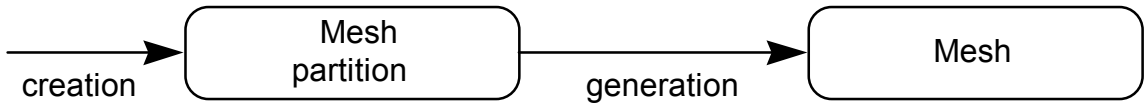
Position in the data flow

The *mesh partition* is used early during the construction stage of the numerical terrain model. The *Partition* is the basis data on which everything else lies.

Thus, the user can quickly create a mesh partition as soon as he has sufficient data to know the topography of the study zones. The user can experiment with different mesh configurations : vary the local density of the meshes as needed, i.e. precision vs calculation time.

The mesh partition generates a *Partition*.

Figure 24 Data flow of the mesh partition



Direct links

- The *mesh partition* is always linked to a *Coordinate system* to locate it in space.

Indirect links

- The *Partition* generated from the *mesh partition* maintains a link with the latter.
- The *Boundary conditions partition* is created by linking with a *mesh partition* since it uses the mesh skin to set its geometry. The *Boundary conditions* can only be applied on the *meshes* generated by this *mesh partition*.

Topography partition

The *topography partition* is a component presenting all the features of the generic *Partition* with, in addition, a data support allowing the transfer of the topography generated by terrain data on the designated mesh. The *topography partition* interacts with the *Topographic terrain* and with equations modifying the morphology of the domain to produce a new series of topographic values transferred on a designated *Mesh*. The type of data generated is known as a *Scalar field - topography*.

With the *topography partition*, MODELEUR can transfer the terrain topography, with the possibility of local modifications through equations, by generating a new *Scalar field - topography* on the designated *mesh*. This new data will become an input data for the simulator HYDROSIM.

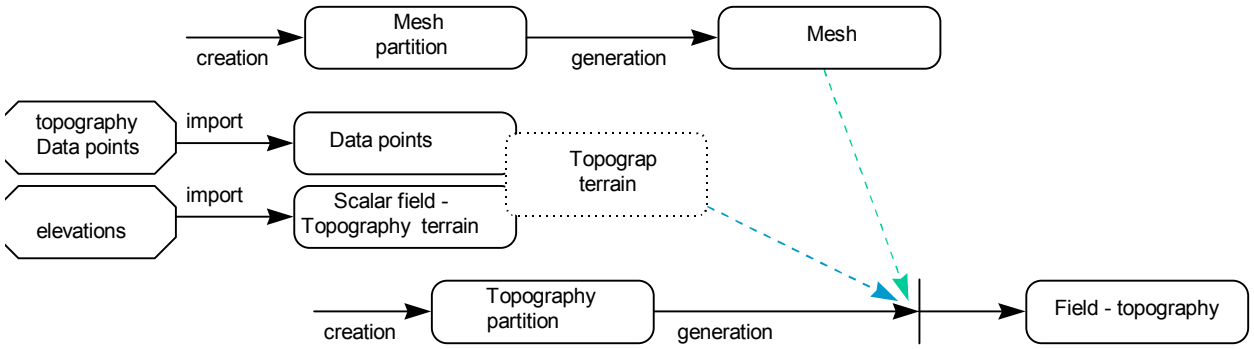
Position in the data flow

The *topography partition* plays a role mainly during the construction of the numerical terrain model. The topographic data generated by field measurements must have been previously imported. The importation of these data will ultimately produce a *Topographic terrain*.

In parallel with this work, the conception of the *Mesh(es)* dedicated to the hydrodynamic calculation must have been initiated. The *meshes* are generated from the *Mesh partition* accessible in the mesh generation mode of MODELEUR.

Finally, the user can construct a *topography partition* with which he can specify the source of the topographic data for the different zones of the study domain. When everything is ready, the user can transfer the topography on one or more *meshes* of hydrodynamic calculation. The *Scalar field - topography* becomes an input data for the *numerical simulation*.

Figure 25 Data flow of the topography partition



Direct links

- The *topography partition* is always linked to a *Coordinate system* to locate it in space
- The *topography partition* may refer to one or more *Topographic terrain*.

Indirect links

- The *Scalar field - topography* refers to the *topography partition* from which it was generated.

Boundary conditions partition

The *boundary conditions partition* is a component presenting all the features of the generic *Partition* with, in addition, a data support allowing the specification of *boundary conditions*.

The *boundary conditions partition* is linked to a *Mesh partition* at the time of its creation. Thus, the skin of the study domain can be automatically generated. The skin is the contour of the domain and of the islands which may be present within the domain. However, the user cannot modify the geometry of the partition since it is generated from the *mesh partition*.

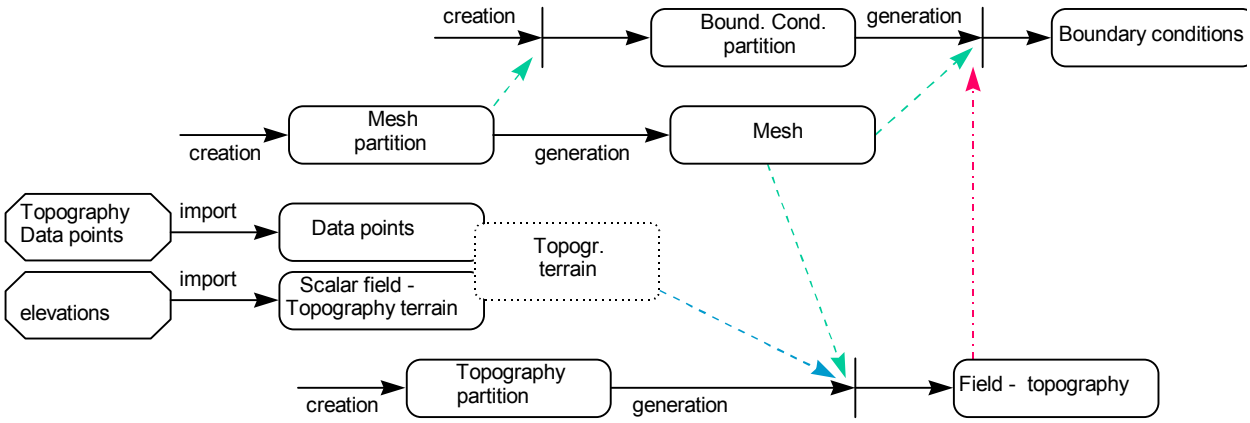
With the *boundary conditions partition*, MODELEUR can generate *Boundary conditions* required for the hydrodynamic simulation.

Position in the data flow

The *boundary conditions partition* is used all along the preparation of the numerical terrain model to launch *hydrodynamic simulations*. The user can return frequently to the partition to generate different hydrological events: normal flow, flood, extreme flood, low water...

The *boundary conditions partition* generates *Boundary conditions* on a compatible mesh. Topography data may be needed to evaluate the distribution of water input on a transect, if the discharge has been specified as a volume (m^3/s). This explains the link with a *Scalar field - topography* in Figure 26.

Figure 26 Data flow of the boundary conditions partition



Direct links

- The *boundary conditions partition* is always linked to a *Coordinate system* to locate it in space
- The *boundary conditions partition* is always linked to a *Mesh partition* from which it generates its geometry.

Indirect links

- The *Boundary conditions* generated from the *boundary conditions partition* maintain a link to the latter.

Substrates partition

The *substrates partition* is a component presenting all the features of the generic *Partition* with, in addition, a data support allowing the specification of the proportions of substrates. To create a *substrates partition*, it is essential that a *Substrate classes,* defining the categories of substrates of the river bed as observed in field surveys, has been previously imported.

The user can then specify, by sub-domain, the proportions of each substrate category. It is also possible to interact with a *Substrates terrain* for a sub-domain allowing to use terrain data specified in the form of *data points*.

Using the *substrates partition,* a *Scalar field - substrate friction* can be generated, comprising a friction coefficient on all points of the mesh selected. The friction coefficient is calculated from the proportions of substrates specified in the partition.

Position in the data flow

The *substrates partition* plays a role mainly during the construction of the numerical terrain model. A *Scalar field - substrate friction* can be generated on a designated mesh (See Figure 21 data flow for substrates).

In parallel with this work, the conception of the *Mesh(es)* dedicated to the hydrodynamic calculation must have been initiated. The *meshes* are generated from the *Mesh partition* accessible in the mesh generation mode of MODELEUR.

The *Scalar field - substrate friction* becomes an input data for the *Hydrodynamic simulation.*

Direct links

- The *substrates partition* is always linked to a *Coordinate system* to locate it in space
- The *substrates partition* is always linked to a *Substrate classes*.
- If the *substrates partition* has used a *Substrates terrain*, it will be linked to the latter.

Indirect links

- The *Scalar field - substrate friction* maintains a link to the *substrates partition* from which it was generated.

Wind partition

The *wind partition* is a component presenting all the features of the generic *Partition* with, in addition, a data support allowing the specification of the constraints related to wind. The *wind partition* specifies, at the sub-domain level, wind strength and direction.

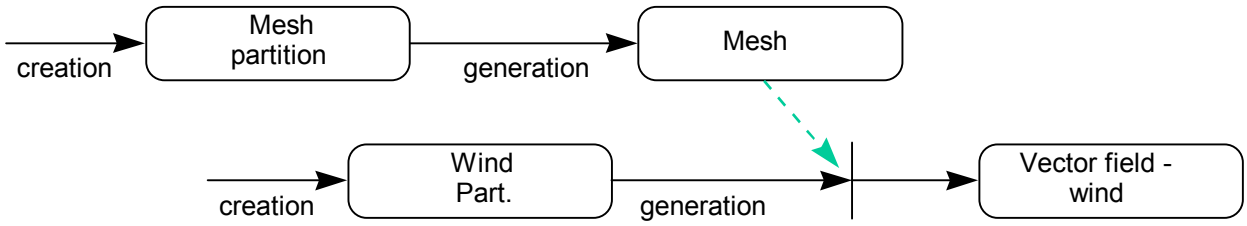
Using the *wind partition*, MODELEUR can produce a *Vector field - wind* transferring the specifications on each node of the designated mesh.

Position in the data flow

The *wind partition* plays a role mainly during the construction of the numerical terrain model. A *Vector field - wind* can be generated on he designated mesh.

In parallel with this work, the conception of the *Mesh(es)* dedicated to the hydrodynamic calculation must have been initiated. The *meshes* are generated from the Mesh partition accessible in the mesh generation mode of MODELEUR.

Figure 27 Data flow for wind



The *vector field - wind* becomes an input data for the *Hydrodynamic simulation.*

Direct links

- The *wind partition* is always linked to a *Coordinate system* to locate it in space

Indirect links

- The *vector field - wind* maintains a link with the *wind partition* from which it was generated.

Ice partition

The *ice partition* is a component presenting all the features of the generic *Partition* with, in addition, a data support allowing the specification of the criteria of friction and thickness related to ice on the sub-domains of the partition.

The criteria to be specified on a sub-domain are the ice thickness and the friction associated with this type of ice. Friction is a Manning coefficient with a value ranging from 0.00 and 0.05.

Using the *ice partition* in the ice mode, MODELEUR can transfer these features on each node of the designated mesh by creating a *Field - ice*.

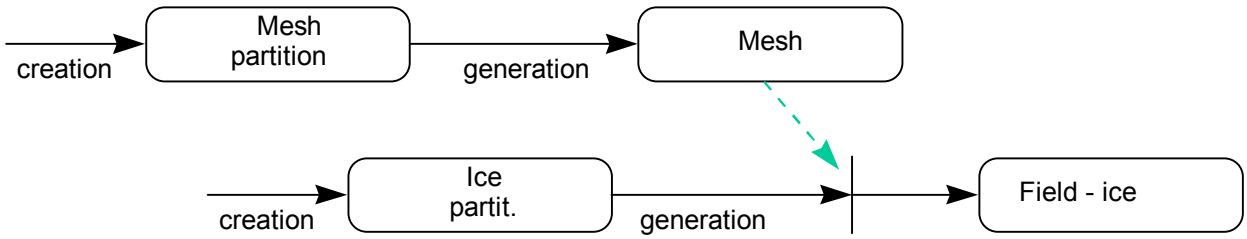
Position in the data flow

The *ice partition* plays a role mainly in the construction of the numerical terrain model. A *Field - ice* can be generated on a designated mesh.

In parallel with this work, the conception of the *Mesh(es)* dedicated to the hydrodynamic calculation must have been initiated. The *meshes* are generated from the *Mesh partition* accessible in the mesh generation mode of MODELEUR.

The *field - ice* becomes an input data for the *Hydrodynamic simulation*.

Figure 28 Data flow for the ice



Direct links

- The *ice partition* is always linked to a *Coordinate system* to locate it in space

Indirect links

- The *Field - ice* maintains a link with the *ice partition* from which it was generated.

Manning friction partition

The *Manning friction partition* is a component presenting all the features of the generic *Partition* with, in addition, a data support allowing the specification of the friction criteria on the sub-domains of the partition. The friction criteria which can be specified on this type of partition is a manning coefficient on each sub-domain of the partition. The values admissible for the coefficient should be between 0.00 et 0.05.

Using the *Manning friction partition*, MODELEUR can transfer on a *Scalar field - Manning* the friction associated with each node of the *Mesh* to which it is linked.

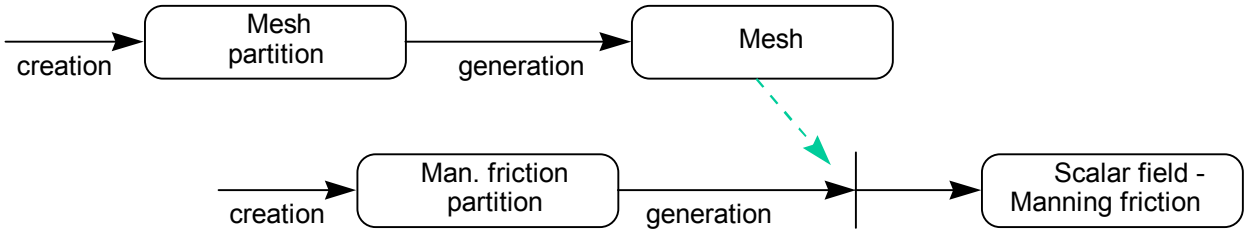
Position in the data flow

The *Manning friction partition* plays a role mostly during the construction of a numerical terrain model. A *Scalar field - Manning* can be generated on a designated mesh.

In parallel with this work, the conception of the *Mesh(es)* dedicated to the hydrodynamic calculation must have been initiated. The *meshes* are generated from the *Mesh partition* accessible in the mesh generation mode of MODELEUR.

The *Scalar field - Manning* becomes an input data for the *Hydrodynamic simulation*.

Figure 29 Data flow for the Manning friction



Direct links

- The *Manning friction partition* is always linked to a *Coordinate system* to locate it in space

Indirect links

- The *Scalar field - Manning* refers to the *Manning friction partition* from which it was generated.

Coordinate system

The *coordinate system* describes the nature of the spatial representation of a spatial data. In the software, the data must be localized in space. All the *Data points,* all the *Mesh(es)* and all the *Partition(s)* must specify in which coordinate system they are described.

The user has all the flexibility needed to integrate data from various sources. The following coordinate systems are available :

- Cartographic coordinate system (rectangular systems : UTM, MTM...).
- Geographic coordinate system (longitude/latitude)
- Cartesian coordinate system (rectangular system without any notion of georeference)

The software now have all the information to represent data from various sources in the same representation space.

Position in the data flow

A *coordinate system* cannot be created manually in the software. The system must be specified in the prescribed format and imported in the database. (For more information on the format of the coordinate systems, see the documentation on data format).

Direct links

- The *cartesian coordinate system* is never linked to other data.
- The *cartographic and geographic coordinate systems* must always be linked to the current project because it contains the coordinate system of the internal system. A coordinate system must always be linked to the internal system to be able to transform spatial data of an external coordinate system to the internal coordinate system, and conversely.

Indirect links

- Data with spatial references always refer to a coordinate system : *Data points*, *Mesh*, *Partitions*...

Data points

To integrate terrain data such as topography, you generally have the use of *data points* composed of a series of measurement points located in space. Associated to these *data points*, there are one or more value files carrying information on all the points of the *data points*.

The *data points* are a special type of *Mesh* in that they are *data points* only for a brief period, because they are immediately transformed into a complete mesh when integrated to the data base.

To import the *data points*, you must use the filter dedicated to the importation of data points, which accepts a series of georeferenced nodes. When loading for the first time, these *data points* are transformed in a complete *mesh* when a Delaunay triangulation

is applied to create T3 elements. Terrain data related to these *data points* can then be viewed.

The *data points* must also be linked to a *Coordinate system* to locate them in space.

Position in the data flow

The *data points* are widely used in the terrain data integration phase and in the construction phase of the numerical terrain model. Since these *data points* immediately become a mesh with T3 elements, they can be used to validate data that have been imported in comparison with data generated by MODELEUR.

However, remember that a mesh with T3 elements cannot be used to execute hydrodynamic simulations. Only meshes with T6L elements have the capacity to do so.

Direct links

- *Data points* are always linked to a *Coordinate system* to locate them in space.

Indirect links

- Various nodal values can refer to *data points*. More specifically:
  - *Scalar field - topography terrain*
  - *Field - percentage of substrates*

Hydrodynamic simulation

The *hydrodynamic simulation* is a central data in MODELEUR since it is the specification of a simulation to be transmitted to HYDROSIM. In the simulation console, a simulation is created and modified by specifying the data to be used for the simulation and by specifying the initial conditions.

Topographic terrain

The *topographic terrain* is an aggregate of two types of data defining it : the *Data points* and the *Scalar field - topography terrain*. This type of data is used in MODELEUR in the topography transfer mode.

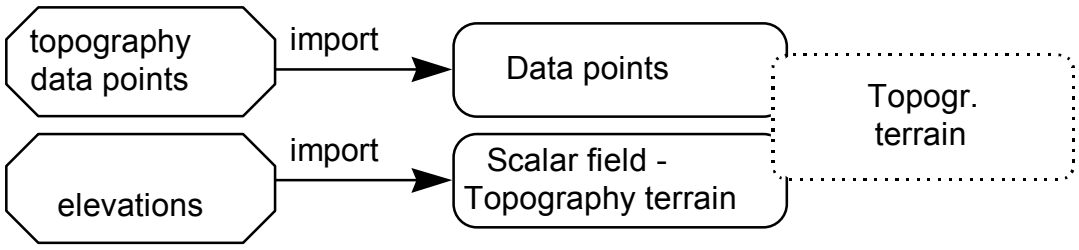
The *topographic terrain* is automatically generated at the time of the importation of the *Scalar field* - topography terrain and of the *Scalar field - topography terrain* (See Figure 30). It is actually the representation of the topographic terrain on a given zone.

The user can then use various *topographic terrains* to cover the entire study domain. The user does not have to select *Data points* and a *Scalar field - topography terrain*, he simply selects the desired *topographic terrain.*

Position in the data flow

The *topographic terrain* cannot be created manually ; it is automatically generated by the database during the importation of terrain data. After importing *Data points,* a *Scalar field - topography terrain* is then imported. At the same time, a *topographic terrain* is created.

Figure 30 Creation of a topographic terrain



This data is used in the construction of a *Topography partition* in order to transfer the topography generated by terrain data onto the calculation mesh generated by MODELEUR. In the *topography partition*, a *topographic terrain* is associated to each sub-domain. A single partition may refer to several distinct *topographic terrains.*

Direct links

- The *topographic terrain* always refers to *Data points* (or even to a *Mesh*) and to a *Scalar field - topography terrain*.

Indirect links

- A Topography partition may refer to *topographic terrains*.

Substrates terrain

The *substrates terrain* is an aggregate of two types of data defining it : the *Data points* and the *Field - percentage of substrates*. This type of data is used in MODELEUR in the substrate mode.

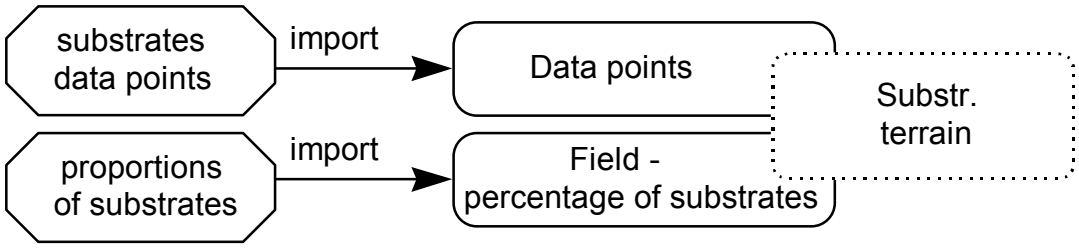
The *field - substrates* is automatically generated at the time of the importation of the *Scalar field* - topography terrain and of the *Field - percentage of substrates* (See Figure 31). It is actually the characterization of the substrates on a given zone, using the related *Substrate classes* to define substrate categories.

The user can then use various *substrates terrains* to cover the entire study domain. The user does not have to select *Data points* and a *Field - percentage of substrates*, he simply selects the desired *substrates terrain*.

Position in the data flow

The *substrates terrain* cannot be created manually ; it is automatically generated by the database during the importation of terrain data. After importing *Data points,* a *Field - percentage of substrates* is then imported. At the same time, a *substrates terrain* is created.

Figure 31 Creation of a substrates terrain



This data is used in the construction of a *Substrates partition* in order to calculate the friction coefficient related to substrates on the calculation mesh generated by MODELEUR. In the *Substrates partition*, a *substrates terrain* is associated to each sub-domain. A single partition may refer to several distinct *substrates terrains*. It is possible to specify directly the proportions of substrates by sub-domain without using the *substrates terrains*.

Direct links

- The *substrates terrain* always refers to *Data points* (or even to a *Mesh*) and to a *Field - percentage of substrates*.

Indirect links

- A *Substrates partition* may refer to the *substrates terrains*.

Chapter 10 Data formats

To import data from sources external to the database, certain format constraints, related to each different type of data, must be respected. Following is the list of formats for each type of data accessible through the data import mechanism :

- Raw topography data format
- Data points format
- Mesh format
- Scalar field format
- Vector field format
- Coordinate system format
  - Cartographic coordinate systems
  - Geographic
  - Cartesian
- Function format
- Substrate classes format
- Mesh partition format
- Topography partition format
- Wind partition format
- Substrate partition format
- Ice partition format
- Friction partition format

Raw topography data format

To import **topography data**, you can import them in the form of raw topography data.

This data contains the following information :

1. Specification of the coordinate system (See *Coordinate system format)*.
2. The number of values in the file.
3. The list of values in the form of :
   `[x y z error_value_on_z]`

Remember that you must provide at least 3 points to have a valid topography data.

Here is an example of raw topography data with 5 values specified in the coordinate system UTM Zone 18 Nad83:

```
transformation=cartographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:utm18n_nad83
5
6.139788135375976e+05 5.206716101699829e+06 9.46042e+01 0.0e+00
6.144025423736572e+05 5.206741525421142e+06 9.43505e+01 0.0e+00
6.140084745788574e+05 5.206360169494628e+06 9.44388e+01 0.0e+00
6.144788135585784e+05 5.206338983047485e+06 9.45025e+01 0.0e+00
6.148262711791992e+05 5.206533898303985e+06 9.45120e+01 0.0e+00
```

Data points format

To import **data points**, the data must be specified in such a way as to respect the format required for this type of data.

The data points must contain the following information :

1. Specification of the coordinate system (See *Coordinate system format)*.
2. The number of nodes in the file, as well as the number of dimensions. Data points can be specified either in two or in three dimensions, although the third dimension will not be kept.
3. The list of nodes, in the format
   `[number_of_nodes coordinateX coordinateY]`.
   The numbering of the nodes begins at 1.

Here is an example of data points with six nodes in geographic coordinates, i.e. in the form of longitude/latitude:

```
transformation=geographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:iden_nad83
6 2
1 -74.251623 44.038678
2 -74.342343 44.058877
3 -74.234587 44.987650
4 -74.451623 44.238678
5 -74.842343 44.158877
6 -74.334587 44.787650
```

Mesh format

To import a **mesh**, the data must be specified in such a way as to respect the format required for this type of data.

The mesh must contain the following information :

1. Specification of the coordinate system (See *Coordinate system format)*.
2. the number of nodes and elements in the file, as well as the number of dimensions. A mesh can be specified either in two or in three dimensions, although the third dimension will not be kept.

3. The list of nodes, in the format
   `[no_node no_field coordinatesX coordinatesY]`.
   Numbering of the nodes begins at 1.
4. The list of connections between nodes, i.e. the elements. There are 7 categories of elements acceptable: L2 (0), L3 (1), L3L (2), T3 (3), T6L (4), T6 (5), Q4 (6). If there are no elements of a certain type, 0 should appear as the number of elements. The connections should be specified according to the standards of Finite elements, anti-clockwise.

Here is an example of a mesh with 6 nodes and 4 T3 elements in geographic coordinates, i. e. in longitude/latitude.

```
transformation=geographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:iden_nad83
6 4 2
1 1 -74.251623 44.038678
2 2 -74.342343 44.058877
3 3 -74.234587 44.987650
4 4 -74.451623 44.238678
5 5 -74.842343 44.158877
6 6 -74.334587 44.787650
0 0
1 0
2 0
3 4
1 1 2 4
2 1 4 3
3 3 4 5
4 3 6 5
4 0
5 0
6 0
```

Scalar field format

To import a **scalar field**, the data must be specified in such a way as to respect the format required for this type of data.

The scalar field must contain the following information :

1. The number of values in the file.
2. The values one after the other.

Here is an example of scalar field associated to a 6 node mesh:

```
6
123.456
321.654
543.123
765.678
987.567
123.987
```

Vector field format

To import a **vector field**, the data must be specified in such a way as to respect the format required for this type of data.

The vector field must contain the following information :

1. The number of values in the file and the number of dimensions. The data can be specified either in two or in three dimensions, although the third dimension will not be kept.
2. Values in X and Y [and Z], one after the other.

Here is an example of a vector field associated with a 6 node mesh:

```
6 2
123.456 432.333
321.654 555.123
543.123 -123.234
765.678 965.321
987.567 -23.456
123.987 -435.678
```

Coordinate system format

To specify a **coordinate system**, you must provide the information needed to locate in space the coordinates [x,y]. In fact all spatial data will have to be linked to the coordinate system.

It is possible to specify different coordinate systems, whether they are georeferenced or not.

The following coordinate systems are available:

- cartographic (all the geographic cartesian systems : UTM, MTM, SPCS...),
- geographic (all the systems specified in longitude/latitude),
- cartesian (all the cartesian systems without notion of georeference),

Each type of coordinate system must contain specific information, and must be specified in compliance with the appropriate syntax.

Cartographic coordinate systems

The cartographic coordinate systems must specify a projection and a datum (See the documentation on **proj**). The projection can be predefined or not ; the datum is always predefined. The predefined projections and datum are found in the files used by **proj** in the installation sub-diretory `\geodata`.

In that directory, you will find files dedicated to predefined projections with the extension`.pjc, and files dedicated to predefined datum with the extension .dtm`. If you want to us a predefined projection corresponding to the projection **UTM Zone 18 North** based on a predefined datum **North American 1983**

for **Canada**, you will find in the file `canada_utm.pjc` the section corresponding to the projection wanted and in the file `dtm_mol.dtm` the datum corresponding to the predefined datum wanted.

Extracted from file `canada_utm.pjc` :

```
<utm18n_nad83> # utm18n_nad83: UTM Zone 18 Nord (NAD-83)
proj=utm zone=18 ellps=GRS80
no_defs <>
```

Extracted from file `dtm_mol.dtm` :

```
<mol_nar_b> # [mol]North American 1983 - Canada
proj=molodensky
ellps=GRS80
del=0,0,0
<>
```

The text between <...> corresponds to the identifier of the predefined projection or datum.

Here is a first example of a cartographic coordinate system entirely predefined from the files of **proj**, i. e. a projection **UTM Zone 18 North** based on the datum **North American 1983 (NAD-83)** for **Canada** :

```
transformation=cartographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:utm18n_nad83
```

The first line identifies the type of coordinate system, in this case a cartographic system.

The second line identifies the predefined datum wanted. Note the syntax of this line which contains in the case of predefined datum :
`datum=`**`init=`**`nom_fichier:non_identifiant`.
The word `init=` reveals that it is a predefined datum ; at this time, only predefined datum are possible.

The third line identifies the predefined projection wanted. Note the syntax of this line which contains in the case of predefined projections :
`projection=`**`init=`**`nom_fichier:nom_identifiant`.
The word `init=` reveals that it is a predefined projection.

Here is a second example of a cartographic coordinate system, this time using a customized projection instead of a predefined projection. We use the same predefined datum as in the previous example, but with a mercator projection :

```
transformation=cartographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=ellps=GRS80;proj=merc;lon_0=0;lat_ts=45d22’00.00n
```

The first and second lines are the same as in the previous example. The type of coordinate system (cartographic) and the predefined datum (NAD-83) are identified.

However, the third line is particular since we do not use a predefined projection ; the parameters required for the projection wanted are specified. Note the presence of the semicolon (;) as separator between parameters. Are specified the ellipsoid (`ellps=`), the projection (`proj=`), the longitude 0 (`lon_0=`) and the latitude ts (`lat_ts=`). Note that the required parameters may vary between projection types ; therefore, you must refer to the documentation on **proj**.

Geographic coordinate systems

The geographic coordinate systems are a special case in the georeferenced context. A geographic coordinate system, i.e. a system specified in longitude/latitude does not require transformation to be represented in the internal system of the current project, because the internal system directly takes longitudes/latitudes as input to make the transformation toward the internal system (cartographic). Conversely, the internal system directly transforms the coordinates in longitude/latitude toward the external system.

This is why a geographic coordinate system only has to specify a projection identity, i.e. without transformation. However, it must specify a precise datum.

For example, to specify a geographic coordinate system, i.e. longitude/latitude, described for the datum **North American 1983 - Canada**, you only have to add a projection ***identity*** described for the same ellipsoid. Here is an example :

```
transformation=geographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:iden_nad83
```

The first line identifies the type of coordinate system, in this case a geographic coordinate system.

The second line identifies the predefined datum wanted. Note the syntax of this line which contains in the case of predefined datum :
`datum=`**`init=`**`nom_fichier:non_identifiant`.
The word `init=` reveals that it is a predefined datum ; at this time, only predefined datum are possible.

The third line identifies the predefined projection wanted. The third line identifies the predefined projection wanted. Note the syntax of this line which contains in the case of predefined projections :
`projection=`**`init=`**`nom_fichier:nom_identifiant`.
The word `init=` reveals that it is a predefined projection. In this case, we are using a projection identity which was predefined. We could also customize a projection for the identity :

```
transformation=geographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=proj=identity;ellps=GRS80
```

The third line is particular since we are not using a predefined projection ; we must specify the required parameters for the projection wanted. Note the presence of the semicolon (;) as separator between parameters. The projection (`proj=`) and the ellipsoid (`ellps=`) are specified. The projection `identity` is a type of projection acknowledged by **proj**.

Cartesian coordinate systems

Cartesian coordinate systems are used in studies which are not using georeferences, i.e. to positioning somewhere in the world.

To specify a cartesian coordinate system, simply identify the type of coordinate system and specify if there is a translation/rotation : without translation/rotation (0) and with translation/rotation (1).

Here is an example without translation/rotation :

```
transformation=cartesienne
translation/rotation=0
```

When there is a translation/rotation, you must provide two points in the system of origin and two corresponding points in the destination system in the form of ;
```
p1a[x,y] p2a[x,y]
p1b[x,y] p2b[x,y]
```

Here is an example with translation/rotation :

```
transformation=cartesienne
translation/rotation=1
100 100 300 300
0 0 200 200
```

Function format

To import a **function**, the data must be specified in such a way as to respect the format required for this type of data. In this case, we import all the text contained in the file without interpretation.

Here is an example of function:

```
function scalar factoriel (scalar scal_facto)
   cmp_factoriel = 1.0
   factoriel = 1
   if (scal_facto < 0) then
      factoriel = -1
   else
      while (cmp_factoriel <= scal_facto)
         factoriel = factoriel * cmp_factoriel
         cmp_factoriel = cmp_factoriel + 1
      end while
   end if
end function
```

Substrate classes format

Substrate classes must be specified in the format prescribed for these type of data. With a substrate classification, you can define up to seven (7) categories or substrate diameter ranges.

These ranges must represent a continuous scale. The substrate classes takes the form of seven range identifier (name) and of 8 numerical values forming the ranges. The ranges must be specified from the largest substrate to the smallest. The smallest range cannot begin with the value 0.0, but a very small value, for example 1.0^{e}-10.

Here is an example of substrate classes with seven categories :

```
7
Metric boulders
Boulders
Pebbles
```

```
Stones
Gravel
Sand
Fine sand
8
10.0
1.0
0.25
0.064
0.032
0.0004
0.00006
1.0e-10
```

Mesh partition format

The mesh partitions must be specified in the format require for this type of data.

The mesh partition carries information at the level of the vertexes of the partition. With each vertex is associated a mesh parameter containing three information : the average spacing of the major axis, the average spacing of the minor axis and the tilt angle.

The partition contains some characteristic sections :

1. Specification of the **coordinate system** since the partition carries spatial information (See Coordinate system format)
2. The number of vertexes and each vertex on a separate line. In the case of the mesh partition, the format for a vertex is :
   `coord_X coord_Y ecart_grand_axe ecart_petit_axe inclinaison`.
3. The number of edges and each edge on a separate line (the numbers of two vertexes at both ends of the edge). The format is :
   `numero_sommet1 numero_sommet2`.
4. The number of surfaces and each surface following. The surface is specified with :
   - the number of vertexes and all the vertex numbers on the following line,
   - the number of edges and all the edge numbers on the following line.
5. Finally, the partition defines its global parameters. For the mesh partition, we have :
   - Default parameters to be assigned to a vertex, i. e. average spacing of the major axis, average spacing of the minor axis, and tilt angle.
   - The type of element to generate when assembling the mesh. You have three possible elements : T3 (3), T6L(4), T6(5). Note that only type T6L can be used for hydrodynamic calculation.
   - The type of mesh generation algorithm. You have access to two types of generation algorithms : isotropic (0) and anisotropic (1).

Here is an example of a mesh partition :

```
transformation=cartographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:utm18n_nad83
6
614023.75 5206586.5 40 5 0
614325.4375 5206623.5 50 15 0
```

```
614267.8125 5206329.0 50 15 0
614040.6875 5206383.0 30 3 0
614616.9375 5206339.0 30 3 0
614623.75 5206613.5 40 5 0
7
1 2
2 3
3 4
4 1
3 5
5 6
6 2
2
4
1 2 3 4
4
1 2 3 4
4
3 5 6 2
4
5 6 7 2
40.0 5.0 0.0 4 1
```

We have a partition specified in a cartographic coordinate system **UTM Zone 18 North** on the datum **North American 1983** for **Canada**. The partition has 6 vertexes, 7 edges and 2 surfaces. The default parameters are specified ; we want to generate **T6L** elements with the algorithm **anisotropic**. Note that the numbering of vertexes and edges begins at 1.

Topography partition format

The topography partitions must be described in the format appropriate for this type of data.

The topography partition carries information at the level of the surfaces of the partition. With each surface is associated a topography terrain and an equation. However in the import/export format, these information are not taken into account. Thus, if we export a topography partition, we only obtain the geometry of the partition without the information ; when importing, we automatically generate null links with topography terrains and empty equations.

The partition contains some characteristic sections :

1. Specification of the **coordinate system** since the partition carries spatial information (See Coordinate system format)
2. The number of vertexes and each vertex on a separate line. In the case of the topography partition, the format for a vertex is :
   `coord_X coord_Y`
3. The number of edges and each edge on a separate line (the numbers of two vertexes at both ends of the edge). The format is :
   `numero_sommet1 numero_sommet2`.
4. The number of surfaces and each surface following. The surface is specified with :
   - the number of vertexes and all the vertex numbers on the following line,
   - the number of edges and all the edge numbers on the following line.

Here is an example of a topography partition:

```
transformation=cartographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:utm18n_nad83
11
614054.0625 5206556.5
614090.625 5206372.0
614186.0625 5206367.5
614302.1875 5206366.0
614380.125 5206340.5
614372.1875 5206564.5
614466.0 5206316.5
614652.0625 5206241.5
614806.375 5206243.5
614796.8125 5206456.5
614534.375 5206545.5
12
1 2
2 3
3 4
4 5
5 6
6 1
5 7
7 8
8 9
9 10
10 11
11 6
2
6
1 2 3 4 5 6
6
1 2 3 4 5 6
7
5 7 8 9 10 11 6
7
7 8 9 10 11 12 5
```

We have a partition specified in a cartographic coordinate system **UTM Zone 18 North** on the datum **North American 1983** for **Canada**. It has 11 vertexes,12 edges and 2 surfaces.

Wind partition format

The wind partitions must be described in the format appropriate for this type of data.

The wind partition carries information at the level of the surfaces of the partition. With each surface are associated wind parameters. A wind parameter contains a orientation in degrees, an intensity and the intensity unit (knots (0), km/h (1) or m/s (2)).

The partition contains some characteristic sections :

1. Specification of the **coordinate system** since the partition carries spatial information (See Coordinate system format)
2. The number of vertexes and each vertex on a separate line. In the case of the wind partition, the format for a vertex is :
   `coord_X coord_Y`

3. The number of edges and each edge on a separate line (the numbers of two vertexes at both ends of the edge). The format is : `numero_sommet1 numero_sommet2`.
4. The number of surfaces and each surface following. The surface is specified with :
   - the number of vertexes and all the vertex numbers on the following line,
   - the number of edges and all the edge numbers on the following line.
   - wind parameters (orientation in degrees, wind intensity and intensity unit (knots (0), km/h (1) or m/s (2)).
5. The default parameters of the partition, i. e. wind orientation in degrees, wind intensity and intensity unit (knots (0), km/h (1) or m/s (2)).

Here is an example of wind partition :

```
transformation=cartographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:utm18n_nad83
6
614111.625 5206508.0
614292.875 5206509.5
614303.375 5206319.0
614045.6875 5206346.0
614561.0625 5206296.5
614528.0625 5206526.0
7
1 2
2 3
3 4
4 1
3 5
5 6
6 2
2
4
1 2 3 4
4
1 2 3 4
50 3.0 2
4
3 5 6 2
4
5 6 7 2
45 2.0 2
0 0.0 2
```

We have a partition specified in a cartographic coordinate system **UTM Zone 18 North** on the datum **North American 1983** for **Canada**. It has 6 vertexes, 7 edges and 2 surfaces. The default parameters are specified, wind direction at 0 degree, and intensity at 0 m/s. Note that numbering of vertexes and edges begins at 1.

Substrate partition format

The substrate partitions must be described in the format appropriate for this type of data.

The substrate partition carries information at the level of the surfaces of the partition. With each surface is associated a substrate terrain or a specification of the percentage of substrate for the surface. However in the import/export format, these information are not taken into account. Thus, if we export a substrate partition, we only obtain the geometry of the partition without the information ; when importing, we automatically generate null links with substrate terrains and default specifications for the percentage of substrate.

The partition contains some characteristic sections :

1. Specification of the **coordinate system** since the partition carries spatial information (See Coordinate system format)
2. The number of vertexes and each vertex on a separate line. In the case of the substrate partition, the format for a vertex is :
   `coord_X coord_Y`
3. The number of edges and each edge on a separate line (the numbers of two vertexes at both ends of the edge). The format is :
   `numero_sommet1 numero_sommet2`.
4. The number of surfaces and each surface following. The surface is specified with :
   - the number of vertexes and all the vertex numbers on the following line,
   - the number of edges and all the edge numbers on the following line.

Here is an example of a substrate partition:

```
transformation=cartographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:utm18n_nad83
11
614054.0625 5206556.5
614090.625 5206372.0
614186.0625 5206367.5
614302.1875 5206366.0
614380.125 5206340.5
614372.1875 5206564.5
614466.0 5206316.5
614652.0625 5206241.5
614806.375 5206243.5
614796.8125 5206456.5
614534.375 5206545.5
12
1 2
2 3
3 4
4 5
5 6
6 1
5 7
7 8
8 9
9 10
10 11
11 6
2
6
1 2 3 4 5 6
6
1 2 3 4 5 6
```

```
7
5 7 8 9 10 11 6
7
7 8 9 10 11 12 5
```

We have a partition specified in a cartographic coordinate system **UTM Zone 18 North** on the datum **North American 1983** for **Canada**. It has 11 vertexes,12 edges and 2 surfaces.

Ice partition format

The ice partitions must be described in the format appropriate for this type of data.

The ice partition carries information at the level of the surfaces of the partition. With each surface are associated ice parameters. An ice parameter contains an ice thickness and a Manning friction coefficient.

The partition contains some characteristic sections :

1. Specification of the **coordinate system** since the partition carries spatial information (See Coordinate system format)
2. The number of vertexes and each vertex on a separate line. In the case of the ice partition, the format for a vertex is :
   `coord_X coord_Y`
3. The number of edges and each edge on a separate line (the numbers of two vertexes at both ends of the edge). The format is :
   `numero_sommet1 numero_sommet2`.
4. The number of surfaces and each surface following. The surface is specified with :
   - the number of vertexes and all the vertex numbers on the following line,
   - the number of edges and all the edge numbers on the following line.
   - the ice parameters (ice thickness in meters, and Manning friction coefficient)
5. The default parameters of the partition, i. e. ice thickness and Manning friction coefficient.

Here is an example of an ice partition:

```
transformation=cartographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:utm18n_nad83
6
613905.0625 5206559.5
614254.25 5206573.0
614250.875 5206281.5
613810.1875 5206312.0
614671.1875 5206264.5
614627.125 5206610.0
7
1 2
2 3
3 4
4 1
3 5
5 6
6 2
```

```
2
4
1 2 3 4
4
1 2 3 4
0.1 0.003
4
3 5 6 2
4
5 6 7 2
0.02 0.005
0.2 0.001
```

We have a partition specified in a cartographic coordinate system **UTM Zone 18 North** on the datum **North American 1983** for **Canada**. It has 6 vertexes, 7 edges and 2 surfaces. The default parameters are specified, ice thickness at 0.2 meters and Manning coefficient at 0.001. Note that numbering of vertexes and edges begins at 1.

Friction partition format

The friction partitions must be described in the format appropriate for this type of data.

The friction partition carries information at the level of the surfaces of the partition. With each surface are associated friction parameters. A friction parameter contains only a Manning coefficient.

The partition contains some characteristic sections :

1. Specification of the **coordinate system** since the partition carries spatial information (See Coordinate system format)
2. The number of vertexes and each vertex on a separate line. In the case of the friction partition, the format for a vertex is :
   `coord_X coord_Y`
3. The number of edges and each edge on a separate line (the numbers of two vertexes at both ends of the edge). The format is :
   `numero_sommet1 numero_sommet2`.
4. The number of surfaces and each surface following. The surface is specified with :
   - the number of vertexes and all the vertex numbers on the following line,
   - the number of edges and all the edge numbers on the following line.
   - the friction parameters, i. e. Manning coefficient
5. The default parameters of the partition, i. e. a Manning friction coefficient.

Here is an example of a friction partition:

```
transformation=cartographique
datum=init=dtm_mol.dtm:mol_nar_b
projection=init=canada_utm.pjc:utm18n_nad83
8
613959.3125 5206505.0
614247.4375 5206569.5
614508.5 5206552.5
614528.8125 5206464.5
614240.6875 5206464.5
614044.0625 5206274.5
```

```
614284.75 5206257.5
614694.9375 5206223.5
10
1 2
2 3
3 4
4 5
5 6
6 1
6 7
7 5
7 8
8 4
3
6
1 2 3 4 5 6
6
1 2 3 4 5 6
0.004
3
6 7 5
3
7 8 5
0.005
4
7 8 4 5
4
9 10 4 8
0.001
0.01
```

We have a partition specified in a cartographic coordinate system **UTM Zone 18 North** on the datum **North American 1983** for **Canada**. It has 8 vertexes, 10 edges and 3 surfaces. The default parameters are specified, and Manning coefficient at 0.01. Note that numbering of vertexes and edges begins at 1.

Chapter 11 Programming language for the calculator

The calculator establishes an interpreted programming language, particularly adapted to calculations on fields. This language offers the possibility to make calculations on simple variables, as well as on fields of the project.

The documentation on the programming language is divided in several sections:

- Syntax
- Key words
- Symbols
- Basic types
- Expression
- Operators
- Control structures
- Functions

Syntax

The instructions of a program are usually written one per line, but a given instruction can be separated on several lines when the \ symbol is used at the end of the line. They will be interpreted in the order in which they are written.

Each instruction comprises tokens separated by single spaces, tabulation spaces and line ends. The calculator does not distinguish between upper and lower case letters, but it does not restrain the use of either one.

Commentaries can be added to a program. They have no influence on the execution or the results. A commentary is introduced by **#** or ! and ends with the line.

Example :

```
pi = 3.1415# A commentary
# Another commentary
deuxPi = 2 * pi
```

Key words

The **key words** are words reserved by the system and cannot be used as *Symbols*. Here is the list of all the key words of the language :

- abs, acos, and, as, asin, atan
- const, continue, cos
- ddx, ddy, declare, dim
- else, end, exit, exp
- false, fix, for, function
- goto
- if, imp, in, int, integer, intg
- lib, ln, log, logical
- max, min, mod
- next, not
- or
- real
- savefield, scalar, scalarfield, sin, sqrt, step, string
- tan, tensor, tensorfield, then, to, true
- vector, vectorfield
- while, with
- xor
- .x, .y

Symbols

A **symbol** is identified by its name. The name must respect the following rules :

- Cannot be one of the *Key words*
- Have a non null length
- Begin with a letter (upper or lower case), or by the character _ (underscore)
- Each subsequent letter is either a lower case letter, an upper case letter, a number, or the character _ (underscore).

Example :

```
pi = 3.1415# OK
deuxPi      # OK
_deux_PI    # OK
2Pi         # Invalid, begins with a number
```

The length of a symbol name is not limited. Two symbols which differ only by upper case letters versus lower case are considered identical, .

Basic types

Each symbol represents data and has the type of these data. The types supported by the calculator are :

- **scalar** : represents a value in floating point.
- **scalarfield** : represents a scalar field such as a field of depth or temperature.
- **vector** : represents a 2 dimension vector, each dimension being a scalar; velocity at one point.
- **vectorfield** : represents a vector field such as a field of velocity or flux.
- **logical** : represents a logical value *true* or *false*.
- **integer** : represents an integral value.
- **string** : represents a chain of characters

Expression

An **expression** is a sequence of *Operators* and of operands which specify a calculation. The expressions are evaluated according to a regrouping convention (associativity) and to rules of precedence of the *Operators*. Two types of expressions can be specified :

- *Arithmetic expression*
- *Logical expression*

Arithmetic expression

An arithmetic expression is an *Expression* in the form of :

```
value_1   arithmetic_operator    value_2
```

where *valuer_1* and *value_2* can be a constant, a variable, a function call, or another arithmetic expression.

Examples :

```
x = 1.0
x +  max(5.0, sqrt(x + 4))  *  2
```

valuer_1 must be of a level equal or superior to the type of *value_2 :*

Examples :

```
2 +  3        # OK
2.0 + 3.0     # OK
2.0 + 3       # OK
2 + 3.0       # OK  result is a scalar
z = {1,2}
2 * z         # OK
z * 2         # OK
```

The result of an arithmetic expression is a value of the same type as *value_1.*

Logical expression

A simple logical expression is an *Expression* in the form of :

```
expr_scalar oper_logical expr_scalar
```

the result of which is a logical value *true* or *false*. These expressions are used in control structures such as *if - else if - else - end if* and loops *while - end while*.

Example:

```
if (t < 3) then
   t = t + 2
end if
```

A complex logical expression can be formed by combining several simple logical expressions using the operators *and* and *or.*

Example:

```
if (x <> 0 or (t > 0 and t < 12) )
   t = t + 2
end if
```

Operators

The **operators** trigger an operation when they are applied to variables and to other parts of an expression.

Associativity and precedence of operators :

()	from left to right

.x , .y {}	from left to right
^ ^2 ^-1	from right to left
* /	from left to right
+ -	from left to right
>= > <= <	from left to right
== <>	from left to right
=	from right to left
,	from left to right

The logical operators are divided in two groups:

- Relation operators whose associativity is from left to right :

>=	Greater or equal to
>	Greater than
<=	Smaller or equal to
<	Smaller than

- The equality operators whose associativity is from left to right :

==	Equal to
<>	Different from

()

The parentheses **(...)** are used to group the terms of a sub-expression.

Example:

```
1 + (2*3)
```

The parentheses **(...)** are used to introduce a call to a *Function*. In this case, the syntax of the call is :

```
Symbol (<list_of_arguments>)
```

where *list_of_arguments* is a list of expressions separated by commas which represent the parameters of the function. The value of the expression of the function call is given by the return value of the function.

Example :

```
a = sin(0.5*pi)
```

.x , .y

The operators **.x** and **.y** are used respectively to extract in a scalar form the component in x and in y of a vector or a vectorfield.

Examples :

```
x = vect.x  # x contains the component x
y = vect.y  # y contains the component y
```

{}

The operator **{... , ... }** is used to create a vector from two expressions whose result is a scalar, or a vectorfield from two expressions whose result is a scalarfield.

Example :

```
v = {x, y}
```

Control structures

The **control structures** are used to direct the flux of execution of the instructions of a program. Key words specific to each type of structure allow to define blocks of instructions on which the control is applied. The programming language of the calculator offers the following control structures :

- *if - else if - else - end if*
- *if - else*
- *for - end for*
- *while - end while*
- *with - end with*

if - else if - else - end if

The control structure **if - else if - else - end if** allows to execute instructions if a condition or its alternatives are verified. Alternatives may or may not exist for the condition.

Syntax :

```
if (logical_expression)
   list_of_expressions
else if (logical_expression) # Can be repeated
 list_of_expressions
else                          # Else optional
 list_of_expressions
end if
```

Example :

```
if (t <= t0)
   alfa = 0
else if (t >= tf)
   alfa = 1
else
   alfa = (t-to) / (tf-t0)
end if
```

if - else

A more simple control structure **if - else**, written on a single line, can be used when there is only one instruction to execute, if the condition or an eventual alternative is verified. This stucture does not require a *end if*.

Syntax :

```
if (logical_expression) then expression else
expression

#The entire expression must be on the same line
```

Example :

```
if (max>M) then max=M else max=0
```

for - end for

The control structure **for - end for** induces an execution loop. The loop is executed until the increment reaches its upper limit. The increment step is given by the value **step** which, if not specified, is 1 by default.

Syntax :

```
for incr=val_ini to val_fin step val_pas
    list_of_expressions
end for
```

Example :

```
sum=0
for iter=1 to 10 step 1    #step optional if=1
   sum=sum+1
end for
#sum is equal to 10

for iter=1 to 10 step 2
   sum=sum-1
end for
#sum is equal to 5

for iter=1.0  to 10.0 step 2
   sum=sum+1
end for
#sum is equal to 10
```

while - end while

The control structure **while - end while** implants an execution loop. The loop is executed as long as the logical expression is evaluated as true.

Syntax :

```
while (logical_expression)
   list_of_expressions
end while
```

Example :

```
sum = 0
while (t < tf)
   sum = sum + 1
   t = t + 0.5
end while
```

with - end with

The control structure **with - end with** is a calculation loop which allows to make calculations on fields (scalarfield or vectorfield) point by point. The elements of the list of elements are pointers in the fields and give access to the values of the field at a given point. The loop ends when scanning of the field is completed.

Syntax :

```
with (list_of_elements)
   list_of_expressions
end with
```

where list_of_element is a sequence separated by elements commas whose syntax is :

```
variable_scalar in variable_scalarfield
```

or

```
variable_vector in variable_vectorfield
```

Example:

```
with (a in aField, b in bField, c in cField)
   c = a + b
end with
```

Functions

A **function** is a piece of independent code which can be parameterized and re-used at will. It usually represents an algorithm or part of an algorithm. Requesting a function induces the execution of the code forming it by using the parameters provided with the request (see *Function*).

In addition to *Predefined functions*, user functions can be written for specific needs (see *Definition of a user function*)

Definition of a user function

The syntax of a function is :

```
function type_of_return name_of_function(param)
   list_of_instructions
end function
```

The function may return a value of the type **type_of_return**. This is done by assigning a value to **name_of_function**.

The list of parameters **param** is a sequence separated by couple commas **type_of_parameter name_of_parameter** where **typeof_parameter** is one of the *Basic types*.

The **list of instructions** represent the body of the function. It comprises instructions which can be used.

Function request

A function is requested by using its name and by providing in ( ) the list of arguments.

Examples :

```
sin(a)      # Calculates the sinus of a
min(a, b)   # Returns the min for a and for b
```

At the request, the parameters of the function are replaced in the same order by the current arguments. The arguments are transferred by reference, i. e. the function receives the variables themselves and not a copy of the variables. Any change to the parameters will therefore be reflected on the arguments.

Example :

```
# Definition of the function double
# which returns a scalar.

function scalar double (scalar v)
   double = 2*v
end function

a = 1       # Assigns 1 to the variable a
a = double(a)    # Function call
...         # Here a is worth 2
```

Predefined functions

The language comprises a certain number of basic functions which can be divided in categories :

General functions :

The language gives you access to basic general functions :

- abs
- fix
- int
- sqrt

Statistical functions :

The language gives you access to basic statistical functions :

- min
- max

Trigonometric functions :

The language gives you access to basic trigonometric functions and their reverse. These functions are defined in radians :

- sin
- cos
- tan
- asin
- acos
- atan

Logarithmic and exponential functions :

- log
- ln
- exp

Calculus functions :

- ddx, ddy
- intg

Read/write functions :

- savefield

abs

The function **abs** returns the absolute value of the argument.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield.

Examples :

```
v1 = abs(-2)    # v1 is worth 2
v2 = abs(-2.2)  # v2 is worth 2.2
v3 = abs(sf)    # v3 contains a scalarfield
which
```

```
                                    # contains the absolute values
                                    # of each of its elements
```

fix

The function **fix** truncates the value in argument. The result is different from the function *int* which evaluates the floor of the past value. In negative values, we obtain the integer superior to the past number, and in positive values, we obtain the integer inferior to the past number.

The types of argument accepted are :

1 - integer

2 - scalar

Examples :

```
v1 = fix(-2)   # v1 is worth -2
v2 = fix(2)    # v2 is worth 2
v3 = fix(-2.2) # v3 is worth -2
v4 = fix(2.2)  # v4 is worth 2
```

int

The function **int** returns the floor to the value in argument. The result is different from the function *fix* which truncates the value in argument. In both negative and positive values, we obtain the integer inferior to the past number.

The types of argument accepted are :

1 - integer

2 - scalar

Examples :

```
v1 = int(-2)   # v1 is worth -2
v2 = int(2)    # v2 is worth 2
v3 = int(-2.2) # v3 is worth -3
v4 = int(2.2)  # v4 is worth 2
```

sqrt

The function **sqrt** calculates the square root of the argument.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield.

Examples :

```
v1 = sqrt(4)   # v1 is worth 2
v2 = sqrt(4.4) # v2 is worth 2.09762
v3 = sqrt(sf)  # v3 contains a scalarfield
```

```
stemming
                # from sf on which we calculated
                # the square root on all the
                # elements.
```

min

The function **min** calculates the minimum of the argument. There are two applications of this function : the function with one argument and the function with two arguments.

Function **min** with one argument

The types of argument accepted are :

1 - scalarfield

2 - vectorfield

Example :

```
v1 = min(sf)  # v1 is a scalar containing the
              # min of the scalarfield.
v2 = min(vf)  # v2 is a vector containing the
              # min of the vectorfield.
```

Function **min** with two arguments

The types of argument accepted are :

1 - string

2 - integer

3 - scalar

4 - vector

5 - scalarfield

6 - vectorfield

Example :

```
v1 = min("to", "ta") # v1 is worth "ta" (string)
v2 = min(1,2)        # v2 is worth 1 (integer)
v3 = min(1.1, 2.2)   # v3 is worth 1.1 (scalar)
v4 = min(vct1, vct2) # v4 is worth the min
between vct1
                     # and vct2 (vector)
v5 = min(sf1, sf2)   # v5 is a scalarfield
                     # containing the min
evaluated
                     # element by element.
v6 = min(vf1, vf2)   # v6 is a vectorfield
                     # containing the min
evaluated
                     # element by element.
```

max

The function **max** calculates the minimum of the argument. There are two applications of this function : the function with one argument and the function with two arguments.

Function **max** with one argument

The types of argument accepted are :

1 - scalarfield

2 - vectorfield

Example :

```
v1 = max(sf)  # v1 is a scalar containing the
              # max of the scalarfield.
v2 = max(vf)  # v2 is a vector containing the
              # max of the vectorfield.
```

Function **max** with two arguments

The types of argument accepted are :

1 - string

2 - integer

3 - scalar

4 - vector

5 - scalarfield

6 - vectorfield

Example :

```
v1 = max("to", "ta") # v1 is worth "to" (string)
v2 = max(1,2)        # v2 is worth 2 (integer)
v3 = max(1.1, 2.2)   # v3 is worth 2.2 (scalar)
v4 = max(vct1, vct2) # v4 is worth the max
between vct1
                     # and vct2 (vector)
v5 = max(sf1, sf2)   # v5 is a scalarfield
                     # containing the max
evaluated
                     # element by element.
v6 = max(vf1, vf2)   # v6 is a vectorfield
                     # containing the max
evaluated
                     # element by element.
```

sin

The function **sin** calculates the sinus of the argument.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield

Notes:

- The input values must be in radian.
- The result is a value between [-1,1]

Example:

```
v1 = sin(1.5708) # (π/2) v1 is worth 1
v2 = sin(0)      # v2 is worth 0
v3 = sin(sf)  # v3 is a scalarfield whose
              # sin has been evaluated on each
              # element.
```

cos

The function **cos** calculates the cosinus of the argument.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield

Notes:

- The input values must be in radian.
- The result is a value between[-1,1]

Example:

```
v1 = cos(1.5708) # (π/2) v1 is worth 0
v2 = cos(0)      # v2 is worth 1
v3 = cos(sf)   # v3 is a scalarfield whose
               # cos has been evaluated on each
               # element.
```

tan

The function **tan** calculates the tangent of the argument.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield

Notes:

- The input values must be in radian.
- The result is a value between $]-\infty,+\infty[$

Example:

```
v1 = tan(0.7854) # (π/4)v1 is worth 1
v2 = tan(0)      # v2 is worth 0
v3 = tan(sf)  # v3 is a scalarfield whose
```

```
                                # tan has been evaluated on each
                                # element.
```

asin

The function **asin** calculates the arcsinus of the argument.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield

Notes:

- The input values must be between [-1, 1].
- The result is a value in radian.

Example:

```
v1 = asin(0)  # v1 is worth 0
v2 = asin(1)  # v2 is worth 1.5708 (π/2)
v3 = asin(sf) # v3 is a scalarfield whose
              # asin has been evaluated on each
              # element.
```

acos

The function **acos** calculates the arccosinus of the argument.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield

Notes:

- The input values must be between [-1, 1].
- The result is a value in radian.

Example:

```
v1 = acos(0)  # v1 is worth 1.5708 (π/2)
v2 = acos(1)  # v2 is worth 0
v3 = acos(sf) # v3 is a scalarfield whose
              # acos has been evaluated on each
              # element.
```

atan

The function **atan** calculates the arctangent of the argument.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield

Notes:

- The input values must be between $]-\infty,+\infty[$.
- The result is a value in radian.

Example:

```
v1 = atan(1)   # v1 is worth 0.7854 (π/4)
v2 = atan(0)   # v2 is worth 0
v3 = atan(sf)  # v3 is a scalarfield whose
               # atan has been evaluated on each
              # element.
```

log

The function **log** calculates the base 10 logarithm.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield

Note:

- The input values must be between $]0,+\infty[$.

Example:

```
v1 = log(1)    # v1 is worth 0
v2 = log(10)   # v2 is worth 1
v3 = log(10.5) # v3 is worth 1.02119
v4 = log(100)  # v4 is worth 2
v5 = log(sf)   # v5 is a scalarfield on which
               # the log was calculated for each
               # elements.
```

ln

The function **ln** calculates the natural logarithm.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield

Notes:

- The input values must be between $]0,+\infty[$.

Example:

```
v1 = ln(2.71828)  # v1 is worth 1
v2 = ln(7.38906)  # v2 is worth 2
v3 = ln(sf)       # v3 is a scalarfield on which
                  # the ln was calculated for
                  # each element.
```

exp

The function **exp** calculates the exponential.

The types of argument accepted are :

1 - integer

2 - scalar

3 - scalarfield

Notes:

- The input values can be between] -∞,+∞[.

Example:

```
v1 = exp(1)  # v1 is worth 2.71828 (e)
v2 = exp(2)  # v2 is worth 7.38906 (e^2)
v3 = exp(sf) # v3 is a scalarfield on which
             # the exp was calculated for each
             # elements.
```

ddx, ddy

The functions **ddx** and **ddy** calculate respectively the derivative in x and in y of the scalarfield used as argument, and return a scalarfield.

```
function scalarfield ddx(scalarfield sf)
```

```
function scalarfield ddy(scalarfield sf)
```

More complex calculations can be based on these two functions. For example, here is the definition of a function which calculates the gradient of a scalarfield :

Example :

```
function vectorfield grad(scalarfield sf)
   grad = { ddx(sf), ddy(sf) }
end function
```

intg

The function **intg** calculates the integral of the field used as argument. It is defined for :

- a scalarfield with scalar as type of return

```
function scalar intg(scalarfield sf)
```

- a vectorfield with a vector as type of return

```
function vector intg(vectorfield vf)
```

Example:

```
# Calculates the integral of the variable depth
val = intg(depth)
```

savefield

The function **savefield** saves a new field in the database. You can save either a scalar field or a vector field.

```
scalarfield savefield(scalarfield, string)
vectorfield savefield(vectorfield, string)
```

the character string will be used to assign a label to the new entity.

Example:

```
# saves a copy of sf
sf1 = savefield(sf, "Scalar field 1")
# saves a copy of vf vf
vf1 = savefield(vf, "Vector field 1")
```